Structural Engineering and Mechanics, Vol. 7, No. 6 (1999) 589-599
DOI: http://dx.doi.org/10.12989/sem.1999.7.6.589

New generation software of structural analysis and design optimization--JIFEX

Yuanxian Gu†, Hongwu Zhang†, Zhenqun Guan‡, Zhan Kang‡, Yunpeng Li‡† and Wanxie Zhong†

Department of Engineering Mechanics, State Key Laboratory of Structural Analysis of Industrial Equipment, Dalian University of Technology, Dalian 116024, China

Abstract. This paper presents the development and applications of the software package JIFEX, a new finite element system which can be used for structural analysis and optimum design by the modern computer hardware and software technologies such as MS Windows95/NT and Pentium PC platforms. The complete system of JIFEX is programmed with C/C^{++} language to make full use of advanced facilities of MS Windows95/NT. In the system, the finite element data pre-processing, based on the most popular CAD package AutoCAD (R13, R14), has been implemented, so that the finite element modeling could be integrated with geometric modeling of CAD. The system not only has interactive graphics facility for data post-processing, but also realizes the real-time computing visualization by means of the Dynamic Data Exchange (DDE) technique. Running on the Pentium computers, JIFEX can solve large-scale finite element analysis problems such as the ones with more than 60000 nodes in the finite element model.

Key words: finite element software; design optimization; MS Windows95/NT; AutoCAD based modeling; computing visualization.

1. Introduction

The finite element method (FEM) is the most powerful numerical method for engineering analysis that progressed rapidly with the development of computer technology during the past decades. Its practical applications in real life engineering are implemented by the various finite element programs, particularly, a number of commercial software packages. Such kinds of commercial FEM software packages, e.g., NASTRAN, ADINA, ANSYS, PAFEC, ASKA, etc. (Brebbia 1982), have been developed and applied extensively in various industrial fields since the 1970s and have become actual industrial standards of engineering computations. The research and development of such kind of commercial FEM software package is an important field of computational mechanics and is closely related to the theoretical research and industrial applications. It has also become a special software industry among the computer and CAD/CAE industries.

Since 1980, many commercial FEM software packages have been developed with newly

† Professor
‡ Lecturer
‡† Assistant

appeared computer technologies. The first generation of them is developed on mainframe computers, and the second generation is moved to 32bit RISC engineering workstations. Beyond the computational facilities, the pre-processing and post-processing on finite element modeling data and user's interface become more and more important. The modern computer technology, represented by the Pentium computers, MS Windows, Windows95 and Windows NT, have provided new hardware and software platforms for the third generation of FEM packages. In recent years, there appears to be new FEM packages based on the platform of MS Windows and Pentium PC. The pre-processing of finite element modeling of these packages is linked with geometric or product modeling of CAD. There is another trend that the commercial FEM software packages, such as NASTRAN, ANSYS and etc., are being enhanced with the facilities of structural design optimization. This is due to the extensive usage of FEM packages and the increasing requirements on the design optimization technique.

The project presented in the paper aims at developing a new generation FEM package on the platforms of MS Windows95, AutoCAD and Pentium personal computers, i.e., the JIFEX. Firstly, the JIFEX integrated finite element packages JIGFEX, DDJ-W (Zhong 1986) and structural design optimization program MCADS (Gu and Cheng 1990), which have been developed at the DUT and applied in Chinese industries since the beginning of 1980s. This software integration is implemented by means of updating the program with the C language. Although there are efforts to replace FORTRAN, the mother language of FEM software packages, for large-scale FEM software, the JIFEX may be the first one generated with the C (Sanal 1994) language. It has been proved that the C Language is very convenient, and necessary at least at present, for the integration of large-scale software on platforms of Pentium, MS Windows and Windows95. An important progress of the JIFEX is its pre-processing of finite element modeling developed on the basis of AutoCAD (R13, R14)- the most popular CAD package. This way, the modeling and analysis of FEM are integrated directly with CAD environment and obtain a more widely developing space due to the huge user family and advanced techniques of the AutoCAD. Not only the interactive graphics and multiple-window user interface, but also the real-time visualized computations have been implemented by means of MS Windows95 and Dynamic Data Exchange (DDE) support. The JIFEX possesses general purpose and powerful capabilities of structural analysis and design optimization, and can solve very large-scale problems on Pentium PC with only 16MB memory. The recent development of new version JIFEX and its applications are presented in this paper.

2. The general description of JIFEX

The integrated software system JIFEX is composed of four subsystems:

(1) the pre-processing subsystem AutoFEM for finite element modeling
(2) the structural finite element analysis subsystem FEANA
(3) the structural design optimization subsystem MCADS
(4) the graphics subsystem GRAFEM for post-processing and real-time visualization of FEM computation.

The subsystems AutoFEM and GRAFEM can also be used as independent programs with interface of data file in unify format. The AutoFEM is embedded into the CAD package (AutoCAD) for the finite element modeling and the automatic producing of FEM analysis data file. The GRAFEM is an interactive graphics program with MS Windows95 interface and graphical facilities serving the FEM computations, such as contour, colour filling, shading, hidden line and hidden surface removal, etc. These two subsystems will be introduced in the following section, along with the features of the JIFEX in structural modeling. FEM analysis and design optimization are presented here.

2.1. Finite element analysis and structural modeling

The structural finite element analysis of the JIFEX is generally applicable to various engineering fields with the following requirements:

- Structural static strength analysis
- Structural free-vibration (frequencies and modes) and dynamic responses (caused by harmonic excitations and arbitrary excitations) analysis
- Global buckling analysis of complicated structures
- Heat transfer and thermal stress analysis
- Contact analysis of 2D and 3D multi-body elastic or elasto-plastic structures
- Structural elasto-plastic stress analysis.

Some featured advantages of structural modeling and analysis for the JIFEX can be briefly described below.

2.1.1. The multi-level substructure method

The typical parts of complicated structures can be modeled by some basic substructure modes, and they can be assembled with the space moving and rotating to compose substructure components, i.e., super elements. The super elements can be used as higher level substructure modes to assemble higher level super elements. This assembling process can be continued level by level to complete the whole structural modeling and there is no limit on the number of levels as long as the computer disk space is enough. This multi-level substructure method is not only very convenient for complicated structure modeling, but also effective to deal with local nonlinear problems, such as contact and elasto-plastic analysis at the highest level.

2.1.2. The parametric variational principle for elasto-plastic structures and contact problems

The parametric quadratic programming algorithm (Zhong and Sun 1988) and a combined programming and iteration algorithm (Zhang *et al.* 1995) derived from parametric variational principle proposed by Zhong (Zhong and Zhang 1998) were implemented in the JIFEX for the elasto-plastic analysis and contact analysis. It has been proved that the methods implemented in the JIFEX are very efficient for the analysis of nonlinear problems such as elasto-plastic analysis, elastic contact analysis or both in large scale structures.

2.1.3. The versatile structural modeling facilities

These facilities include element library, loading cases, boundary and connection condition descriptions. The element library of JIFEX is composed of: bar, beam, membrane, plate, shell, 3D solid brick, axisymmetric solid and shell, spring, honeycomb sandwich plate, and composite laminate plate. The loading types include nodal forces, distributed loads, volume loads, centrifugal load, temperature loads, nonsymmetric loads in symmetric structures, and multiple modes of beam loads. The boundary support and component connection conditions can be described by displacement constraint on each freedom of node and in arbitrary local co-ordinate system. The displacement constraints include status of fixed; pre-assigned value of displacement, and particularly, multi-level master-slave relation. The master-slave relation means that the displacement status of a freedom of slave nodes can be controlled by (i.e., with the same value of) one master node, and defined level by level. This method makes the simulation of complicated structures convenient and easy with finite element model.

2.1.4. The particular management of data area within memory, database structure on computer disk and data input/output between memory and disk

This data management is implemented by means of a particularly designed file system. It provides three principal types of data structures: (1) self-defined files are used to organize data logically; (2) data records are used to store data as common array and are managed by files; (3) manage type records are used to manage data block with fixed length or varying length and sub-level files. The database structure on disk and data input/output are managed automatically by this file system. This kind of data management mechanism is advantageous for the management of non-structured data and disk database, programming and integration of large scale program system, and effective usage of computer memory. It is also a technique making the JIFEX capable of dealing with large scale problems with small computers.

2.2. Structural design optimization

The structural optimization subsystem MCADS possesses general purpose capabilities described with design variables, constrain functions and optimization objectives. Its design variables can be classified into following types:

(1) Size design variable. It includes the cross section area of bar, the thickness of membrane, plate and shell, and the cross section sizes of beam with various shapes. For the beam elements, the design variables can be any geometric parameters of commonly used cross section, for which a library has been built. The program deals uniformly with the design variables and stress calculation of a beam within this library.

(2) Shape design variable. It includes the co-ordinates of special nodes and geometric parameters used in the shape description of structural boundaries, e.g., interpolation parameters and interpolation point positions of curves or surfaces. A programming interface is provided for users to describe any kind of boundaries in the optimum design shape of continuum structures.

(3) Composite design variable. It includes the ply orientation angles and layer thicknesses of laminate and sandwich plates, height of honeycomb core, and material parameters of special composite plates.

The constraint functions of MCADS cover the following structural response behaviours: structural weight, node displacement, node and element stresses, vibration frequencies, dynamic displacement and stress responses, and structural buckling load. The design objective function can be selected from any constraint functions or combined with several constraint functions. This way, the objective of design optimization can be of :

- minimizing structural weight
- reducing structural stress and deformation

- improving structural stiffness
- increasing structural fundamental frequency or adjusting the distribution of a group of vibration frequencies
- minimizing structural dynamic responses
- increasing structural buckling loads.

The solution algorithms of the MCADS for design optimization problems are sequential linear programming (SLP) and sequential quadratic programming (SQP). The basic SPL and SQP algorithms have been improved by the numerical approaches of approximate line search, adaptive move limit, temporary constraint relaxation and feasibility adjustment. These approaches make the optimization solution algorithms robust and stable. The structural sensitivity analysis of the MCADS is to compute the first order derivatives of structural stress, displacement, vibration frequency, dynamic response, buckling load and the second order derivatives of structural weight. According to the type of design variables, the analytic method or semi-analytic method is used in the sensitivity analysis. The semi-analytic method of sensitivity analysis is a key point of the MCADS to implement the generally applicable design optimization by employing facilities of structural modeling and analysis as "black box".

3. Finite element modeling integrated with AutoCAD

The most important recent progress of the JIFEX is its pre-processing subsystem AutoFEM of finite element modeling integrated with the CAD package (AutoCAD, R13, R14). The numerical modeling and data pre-processing of the finite element analysis are very time consuming and difficult, particularly in the case of complicated 3D solid structures. The research on mesh generation algorithms and computerized methods of FEM pre-processing have been active in the past twenty years. The pre-processing is the developmental emphasis of various commerical finite element packages since the start of the CAD technology progress in the last decade. The complete procedure of finite element modeling and data pre-processing are composed primarily of three phases:

(1) geometric modeling and its data description of structures
(2) automatic generation of finite element mesh
(3) automatic generation of description data of the finite element attributes.

The finite element attributes are those data to form complete finite element model with mesh data, such as loads, boundary conditions, material and cross section parameters of elements. Within the CAD environment, the finite element modeling must be combined with the technology of product modeling or geometric modeling and integrated with their packages. This way, the difficult geometric modeling problem, particularly for 3D solids, can be solved easily, and the finite element analysis is possible to be integrated with geometric design within CAD. Furthermore, the facilities of geometric operations and interactive interfaces of CAD packages can be employed for the finite element modeling.

With this idea, we developed a new subsystem of finite element modeling based on the CAD package AutoCAD (R13, R14) and named it AutoFEM. The AutoCAD (Soen 1995) is the most popular CAD software package extensively used in China and the world. Its advanced product Autodesk Mechanical Desktop is very powerful in the solid and surface modeling. The integration with AutoCAD gives the AutoFEM powerful geometric modeling facilities and convenient interactive operation environment to implement a new pre-processing tool of finite element modeling. It will also make structural analysis and design optimization with the JIFEX integrating with the CAD environment and packages accepted by more users. The development of AutoFEM has made use of the ADS – AutoCAD development system (Autodesk I. 1993) as programming tool that provides C programming interface to use all facilities of C language and the data interface to access the AutoCAD database of geometric entities. The employment of the ADS makes the AutoFEM be embedded into the AutoCAD as a user extension, performing as a complete system, and linking with the JIFEX via the interface of generated data files. In this usage approach, the AutoFEM is also an independent program capable of being linked with any other finite element analysis software. The AutoFEM has the following facilities of finite element modelling and data pre-processing:

- geometric modeling using the AutoCAD
- automatic generation of finite element mesh
- interactively generating the attribute data of finite element model
- interactive construction of the frame structure and beam elements
- interactive edition and modification on finite element mesh and attributes
- graphics viewing of finite element model
- output the data files for the finite element analysis.

Firstly, the structural geometric model is constructed by a user in the AutoCAD with all of its utilities of drawing and modeling. Then, the AutoFEM is loaded to transfer the geometric model into a finite element model used for mesh generation. With the AutoFEM, the frame structures can also be constructed interactively and the beam elements are designed with space positions, shape and sizes of cross sections. The substructure concept is used in the AutoFEM for finite element modeling. The whole structure can be divided into several subregions with simpler shapes, and finite element meshes are generated within each subregion respectively.

There are number of algorithms for mesh generation in 3D space, i.e., the mapping algorithm with mapping function extending and pattern-module method for mesh refinement, the free meshing algorithm with node-distributing and element-connecting to generate finally all quadrilateral element mesh, scanning algorithm to generate suriace and solid mesh, surface mesh generation with 3D-mesh facility of AutoCAD. These algorithms have individual advantages suitable to different cases. For instance, the 3D-mesh algorithm is very simple for surface meshing since it makes use of the 3D-mesh entity defined in the AutoCAD, and the scanning algorithm is effective for rotation bodies. Besides, the free meshing algorithm, with node-distributing and element-connecting procedure, and the modified octree algorithms for automatic mesh generation of 3D solids are on progress of tuning. These algorithms, interfaced with the advanced geometric modeling module AMD (Autodesk Mechanical Desktop), will solve the most difficult problem of automatic meshing of any complicated 3D solid structures.

After mesh generation, the finite element analysis attributes, such as loads, displacement constraints of node freedoms, material parameters, cross section properties of elements, can be defined and assigned to the finite element model interactively with the help of pattern-graph menus. The description data of these attributes will be produced by the AutoFEM and associated with nodes, elements, or subregions. The extension data and its operation of entities with the ADS are applied within the attribute data processing. It is a unified geometric entity and attribute data processing. With dialogue windows of load definition and property definition of beam cross section as shown in Figs. 1 & 2, the selection of attribute types and definition of descriptions will become very easy.

The subregions with generated mesh and attribute data can be edited and modified by the

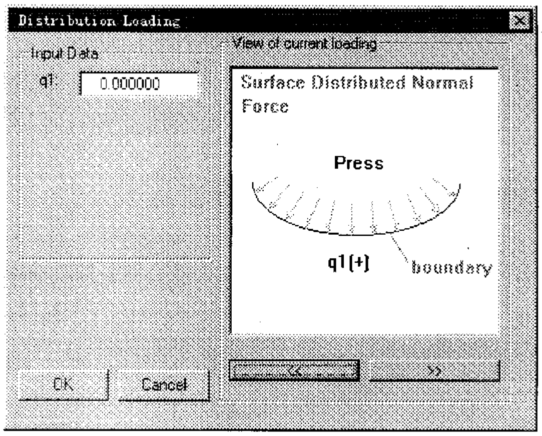

Fig. 1 Dialogue windows of loads

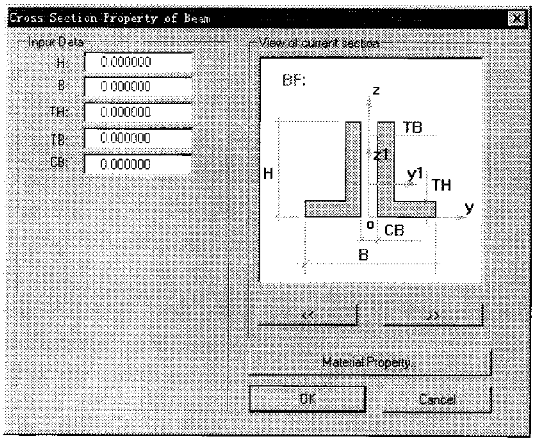

Fig. 2 Dialogue window of beams

interactive operations such as copy, delete, move, rotate, array and mirror. The finite element mesh is capable of being locally modified by moving node positions to improve the mesh quality. Finally, the subregions with generated mesh and attributes are assembled to construct the complicated structural model. The complete data file required for the analysis of the built finite element model is output by the AutoFEM with the JIFEX format and a universal format. The data file in the JIFEX format can be used directly for structural analysis, and the data file in the universal format can easily be transferred into any other format.

Since the AutoFEM is embedded into AutoCAD becoming a component of it, the user interface and usage operations are all the same as the AutoCAD. All of the convenient facilities of geometric modeling, interactive operations and graphics viewing of the AutoCAD have been maintained in the AutoFEM. This point is important for users of the commercial software package. The parameterization design and the feature based modeling of the AutoCAD and AMD can be further employed for the design optimization and adaptive analysis. They can provide a unified general approach and feedback necessary information to CAD design in virtue of CAE computations. On the platform of the MS Windows or MS Windows95, the ADS and new development tool ARX of the AutoCAD are capable of developing programs for large scale application systems integrated with AutoCAD. The benefits of the AutoCAD mentioned before strongly support the development and application of the AutoFEM.

4. Graphics and real-time visualized computing on windows95

The graphics subsystem, also run as an independent program, GRAFEM, is developed and applied on the platforms of MS Windows and MS Windows95. Thus, it has employed the utilities of multiple windows, multiple processes, interactive interfaces, graphics output or hardcopy and etc. to perform a new Windows95 graphics program. The first function of the GRAFEM is data post-processing of finite element analysis with the following computer graphics:

- mesh graph of finite element model and mesh graph with hidden line removal
- shading graphics of 3D finite element model
- mesh graph or shading graphics with structural deformation
- stress or other data distribution expressed by color fillings on body surface, or

- the internal section cut from 3D solid
- computer animation simulating structural vibration or deformation history
- real-time rotating of graph or shading graphics of mesh model.

The second function of the GRAFEM, a particular facility, is the real-time integration of visualization and computation. It means that the computing of finite element analysis and the visualizing of computer graphics are simultaneously running in distributed·processing manner. This is supported by the techniques of multiple processing and dynamic data exchange (DDE) provided by the MS Windows95. The computation of finite element analysis and the visualizations of computer graphics are run within multiple windows, i.e. multiple processes. The result data of analysis computation is sent with particular event message to the processes of graphics visualizations. The graphics visualizations of the GRAFEM run in an event driven waiting loop refreshing their graphics while receiving data and messages during the computation process. Then, a real-time visualized computing with distributed processes and multiple windows is implemented. Compared with static data post-processing, this dynamic real-time computational visualization is a significant progress. It is specially helpful to the iterative computations such as design optimization, time integral, nonlinear iterative analysis, etc., and sound basis of larger scale distributed processing and computational visualization.

5. Practical applications and examples

The JIFEX and its former subsystems, structural analysis program JIGFEX and design optimization program MCADS, have been applied in structural engineering of Chinese industries and they are now on the progress of commercial marketing. In practical applications, it shows some features, such as versatile modeling facilities and multi-level substructure method to simulate complicated structures, effective numerical solution for 3D multi-body contact problems, general applicable design optimization on optimum shape, dynamic properties and composite structures. On Pentium PC, it solved a number of large scale computing problems with 30000-60000 nodes

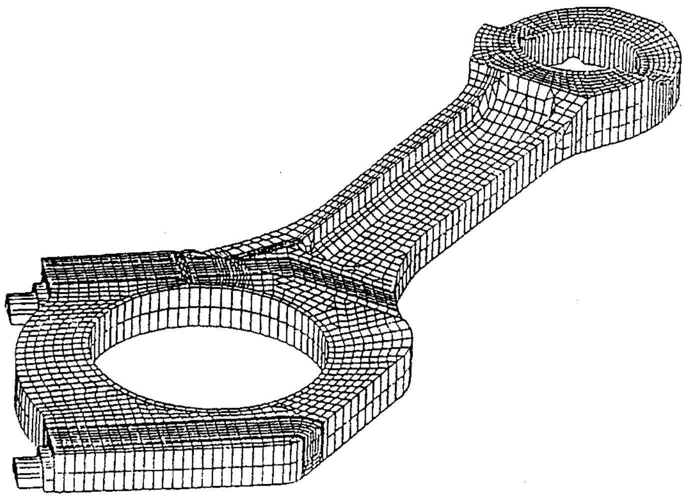

Fig. 3 Finite element model of the 16V240ZJ G rod

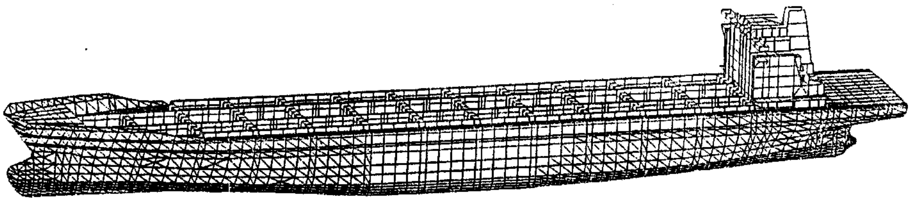

Fig. 4 Finite element model of containership hell structure

in strength analysis and even in contact analysis. This kind of computation was only the job of mainframe computers. Nowadays, more powerful and convenient engineering computations integrated with CAD modeling and graphics visualization can be performed by the hardware of Pentium and the software of MS Windows95 and AutoCAD. On the basis of JIFEX, some special versions have also been developed for the industries of aerospace, shipping, locomotive and diesel. Some analysis examples of locomotive and diesel structures can be seen in Wu (1997). Application examples of structural optimum designs are illustrated below.

5.1. EXAMPLE 1. 3D multi-body contact analysis

This example is about the calculation of 3D connecting rods. The model G rod of the 16V 240ZJ engine is selected and shown in Fig. 3. The finite element model is made up with 8-node bricks. Considering the long-wing in succession and the short-wing in cutting, this model simulates the contact relationship between shell-rod and dolt-rod, and precisely describes the loads exerted on the con-rod in various case loads. The total node number is 9702 and 264 3D contact elements are used. The total degree of freedom is 26000. The example makes an investigation on algorithm accuracy, points out that the design of G rod is rational, and the crack of the short-wing is caused by the minute actual relative moment at the root in various case loads. The tiny moment is caused by the different stiffness between the long-wing and the short-wing. This example also shows the capability of the JIFEX for the analysis of engineering problems.

5.2. EXAMPLE 2. The analysis of complete hull structure of a containership

The finite element model of hull structure of a 50000t containership is shown in Fig. 4. The

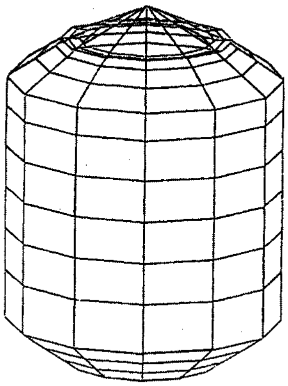

Fig. 5 The finite element model of spaceship orbit-module structure

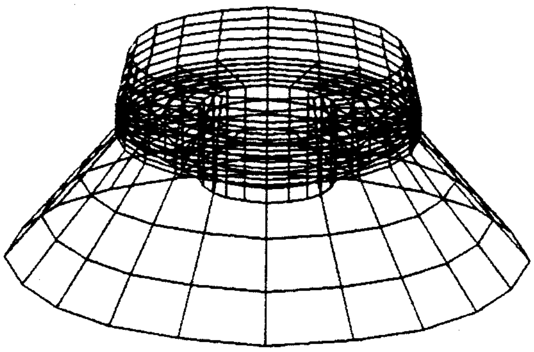

Fig. 6 The finite element model of structural component of carrier rocket CZ-2E

whole geometric model of hull structure is constructed interactively and the finite element mesh is generated automatically with the AutoFEM. The total number of nodes is 10605, the number of elements is 31646, and displacement degrees of freedom are 63594. The case loads along with wave loads and cargo loads are combined in sereval dangerous conditions. On Pentium/120(MHz), the computational time is 4 hours. The computing results of the deformation and the stress distribution are reasonable and agree to design experience.

5.3. EXAMPLE 3. Dynamic property optimization of spaceship orbit-module structure

The shell structure reinforced with beams of spaceship orbit-module shown in Fig. 5 is optimized on the shell thickness and beam cross-section area. By means of optimization, the structural first frequency is increased by 9% with the same weight as original design.

5.4. EXAMPLE 4. Optimum design of structural component of carrier rocket CZ-2E

The finite element model of a structural component of carrier rocket CZ-2E shown in Fig. 6 is composed of shell and beam elements with a number of shapes of cross section. The loading caused by gravity acceleration in the launch stage, and the constraints on the deformation, stress and vibration frequencies are considered in the analysis and optimization. The 12% weight of structure is reduced by the design optimization with only static constraints, and the 8% reduction on the structural weight is obtained with both static and dynamic constraints.

6. Conclusions

It has been illustrated that the JIFEX is one of the new generation software packages of finite element analysis and structural design optimization. The so called new generation FEM software possesses some advanced features such as C/C++ language programming, implementation under the MS Windows95/NT environment, integration of finite element modeling and CAD geometric modeling, interactive graphics with MS Windows interface and real-time concurrent visualizing and computing, and large-scale engineering computing on Pentium PC. These features are supported by the modern technologies of computer hardware and software, and represent the developmental trend of applicable software packages of structural analysis and optimization.

Acknowledgements

The project is supported by the Scientific Fund for National Outstanding Youth of China (19525206) and the National Natural Science Foundation of China (19472014). The authors would like to acknowledge Prof. Wu Changhua who provided the result of the first example. The programming and computing efforts of the students in the authors' group are also appreciatively acknowledged.

References

Autodesk Inc. (1993), *ADS Programmer's Reference.*

Brebbia, C.A. (ed.) (1982), *Finite element systems-A Handbook,* Computational Mechanics Centre Southampton, UK.

Gu, Y.X. and Cheng, G.D. (1990), "Structural shape optimization integrated with CAD environment", *J. Structural Optimization,* **2**(1), 23-28.

Sanal, Z. (1994), "Finite element programming and C", *Computer and Structures,* **31**, 671-686.

Soen, F. (1995), *Inside AutoCAD Release 13 for Windows and Windows NT,* New Riders Publishing.

Wu, C.H. (1997), "Precise analysis of complicate structures in mechanical engineering", *Proc. of CASCM'97,* Sydney, Australia, February.

Zhong, W.X. (1986), *Programming of Computational Structural Mechanics on Microcomputers,* Chinese Hydraulic and Electricity Industrial Press, Beijing.

Zhong, W.X. and Zhang, R.L. (1988), "Parametric variational principles and their quadratic programming solution in plasticity", *Comput. Struct.* **30**, 837-846.

Zhong, W.X. and Sun, S.M. (1988), "A finite element method for elasto-plastic structures and contact problems by parametric quadratic programming", *Int. J. Num. Meth. Eng.,* **26**, 2723-2738.

Zhang, H.W., Zhong, W.X. and Gu, Y.X. (1995), "A combined programming and iteration algorithm for finite element analysis of three-dimensional contact problems", *Acta Mechanica Sinica,* **11**, 318-326.