N87-27234

A NEW APPROACH TO FINITE ELEMENT MODELING, ANALYSIS AND POST-PROCESSING

By: Gil White, Intergraph Corporation

ABSTRACT

Today's structural engineer is confronted with a host of CAD/CAE systems that offer solutions to the problems of finite element model construction, analysis and results interpretation. The history of computer-aided design and analysis has followed a path that provided for the quickest method for automating specific functions. This evolution has created islands of automation. Various attempts have been made to link segregated databases and form cohesive systems. Some attempts have proven sucessful but their efficiency is limited. Recent developments in both hardware and software have opened the door to systems of fundamental new design. The industry is now seeing the development of fully standalone workstation platforms that provide new and exciting opportunities for the COSMIC NASTRAN user.

INTRODUCTION

Increased computer computational power combined with advances in the field of finite element analysis have revolutionized structural engineering. Today, this revolution is continuing and displays itself in the form of full integration between mechanical design and various types of engineering analysis. Systems displaying full functionality between design and analysis are becoming the standard for companies of all sizes as CAE becomes increasingly more cost effective. These systems offer a significant cost improvement by reducing the turnaround time for product development and verification. In many cases, the need for prototype development is omitted from the design process completely.

The ideal finite element modeling system would be a system of multi-functionality that meets the needs of different users. Such a system would have a database that serves as a foundation for geometric design as well as engineering analysis. The COSMIC NASTRAN user is now confronted with more alternatives in the selection of pre- and post-processors as well as methods for full system integration.

A NEW APPROACH TO FINITE ELEMENT MODELING, ANALYSIS AND POST-PROCESSING

The benefits of full integration between CAD and CAE are nowhere more obvious than in the area of finite element analysis. By linking the design and analysis process, the COSMIC NASTRAN user can improve accuracy and reduce modeling time. The cost benefits of such integrated software are multiplied when these functions can be performed on a standalone workstation. The major expense in the area of finite element analysis is incurred during model construction; modeling time is greatly reduced when the analyst uses the geometric design as a basis for the FE model. This concept, known as geometry-based mesh generation, represents the new generation of finite element modeling systems. In addition, integrated design and analysis produces cost savings with respect to trouble shooting, elimination of modeling errors and increased accuracy.

Historically, the COSMIC NASTRAN user has constructed models by one of two methods. He either input all components (nodes, elements and materials) by single entry operations, or he used some type of automated mesh generation process. These methods for inputing information required the engineer to address segments of the model independently; they also required that this data be reformated if the part geometry changed. Both of these limitations increase FEA cost. Therefore, the ideal modeling system would be one that automated these functions by fully integrating design and analysis.

Post-processing and results interpretation are also expedited through graphic and analysis links. Visual inspection of the analysis results provides the quickest method for evaluating the model. This quality of CAE has long been recognized as one of the strong points in its application to the field of finite element analysis. The ideal post-processing environment would not only provide graphic and conventional methods for reviewing output, but would also offer a data management system. Such a data management system would allow the user to manipulate analysis results, to create new output, as well as providing re-analysis capability.

INTERGRAPH addresses the needs of the COSMIC NASTRAN user with a new workstation-based finite element modeling system called I/FEM. This product was developed using object-oriented programming which provides the foundation for true integration between design and engineering analysis. Object-oriented programming methods were chosen over more conventional procedure style programming because it permits code to be reused thus greatly reducing the time dedicated to programming. Also, object-oriented programming allows addition of new data types without modifying existing code.

The I/FEM modeler has the ability to construct all components of the finite element model. Model construction actually begins during the design session when geometry is enhanced with FEA attributes. This "intelligent" geometry greatly accelerates the analysis process. Fully automatic meshing gives fast, accurate and efficient model construction. The user can optimize the mesh with respect to predefined criteria. The system can automatically remesh when required. Loads and boundary conditions may be defined prior to meshing and later edited, or placed at any time after the mesh has been created; the load and boundary condition criteria are assigned to the geometry as opposed to the mesh. The I/FEM modeler constructs the vast majority of input data for the COSMIC NASTRAN user.

The I/FEM solver employs the finite element method in the solution of several classes of applied mechanics and field problems. I/FEM offers basic analysis capabilities that can be expanded by adding other analysis modules. The solver is described as an out-of-core solver which limits model size to the amount of local disk storage only. The I/FEM post-processor offers both graphic and nongraphic methods for reviewing any of the results that are output from the local solver or any third-party solvers including COSMIC NASTRAN. I/FEM offers the ability to do reanalysis within the postprocessing environment. The system will automatically determine if a new decomposition is required and if so will instruct the solver accordingly. Also, within this environment one will be able to perform arithmatic and algebraic operations on any of the output data. New data sets may be created to enhance COSMIC NASTRAN results interpretation. The user-friendly interface of the I/FEM post-processor is seen in the way COSMIC NASTRAN analysis results are reviewed. All naming schemes and nomenclature are transferred from COSMIC NASTRAN to the I/FEM environment without any change. This gives the COSMIC NASTRAN user the feeling of working in one software package.

I/FEM takes an innovative approach to interfacing COSMIC NASTRAN and I/FEM. The ideal data transfer procedure is one which passes all information to a "neutral" file . From this neutral file, routines may be written to access data in a variety of ways. In recognition of the fact that an "open" architecture is essential to a complete system, I/FEM will maintain such a "neutral" file translator. For selected third-party solvers, such as COSMIC NASTRAN, Intergraph will also provide the interface from the neutral file to the third-party solver, thereby completing the interface between I/FEM and COSMIC NASTRAN. This method also provides a reliable way to securely archive models.

CONCLUSIONS

Recent advances in both hardware and software have opened the door to a new generation of finite element modeling systems. INTERGRAPH CORP has combined an innovative programming concept with a standalone workstation hardware platform to produce a new standard in finite element modeling. This system offers the COSMIC NASTRAN user full integration between design and analysis. I/FEM not only addresses the needs of the COSMIC NASTRAN user of today, it also provides for continued evolution of the COSMIC NASTRAN product.