FEM on a PC: Finding the Best Fit

by Ken Klein
Computervision, Personal Systems

ABSTRACT

Advances in computer hardware and software allow the Finite Element Method to be used by a much larger audience. Due to decreasing costs, many companies can now realistically consider placing a personal computer on every engineers desk.

This paper describes some aspects the new user should consider in selecting FEM software solutions. Topics discussed include training, software classification, interactive graphics, and advanced topics.

INTRODUCTION

With the introduction of PC based products in the past two years, FEM has become available to a much larger audience. At the same time, the opportunity for the misrepresentation and misuse of FEM's potential has become much greater. Choosing a PC based product is enough of problem for mainframe FEM veterans. For the group of analysts/designers who are looking into FEM for the first time, it can be mind boggling. How does one shop for FEM software? How does one obtain the knowledge to use the method correctly?

IS FEM FOR YOU?

FEM has become an accepted method for the solution of many different types of problems in acoustics, electromagnetics, thermal and structural analysis. Alternative methods include simplifying assumptions, overdesign, and physical testing of prototypes. The advantages of FEM over other approaches to analysis can probably best be summarized as more accurate, reliable results in less time and the ability to handle structures that are too large, too complex or too expensive to tackle with manual methods.

The user must be adequately trained to use the solution technique correctly. Computer tools have simplified the process so much that inexperienced users can now perform analysis that formerly required experienced analysts. Even with FEM, where the computer "gives you the answer", the solution must be verified by alternative means. FEM training is a combination of classroom education on the fundamentals and experience gained on your own or with a mentor. FEM training is available from a variaty of sources. FEM has now reached the undergraduate level in many colleges and universities. Most of the analysis code vendors provide training on FEM fundamentals in addition to specialized training on the use of their particular analysis code. Before even starting to tackle your own real life analysis problems, you need to understand the basics.

FEM SOFTWARE SELECTION

The first step in selecting FEM software, of course, is a feature by feature comparison. Review the literature in magazines like CAE, get product specific data sheets from vendors, and simply compare capabilities. Then comes the hard part. Say, for example, that you have narrowed the field to four potential products -- all with basically the same capabilities and all the same price, within 25% or so. How do you choose?

You might be tempted to simply choose the one with the lowest price tag, get a pat on the back from Accounting, and be done with it. This is not advisable. FEM products come in quite different flavors, and the one one you choose on a price/performance basis alone may not be palatable to you.

To use a different metaphor, shopping for FEM software is like shopping for an automobile. You can't assume that, because a car has four wheels, an engine, and is within your price range, that it's the right car for your needs. You have to drive it.

TYPES OF FEM SOFTWARE

I have so far used the term "FEM software" rather loosely. The acronym FEM itself sometimes get tossed around too lightly, so let me be clear about what I mean by this term. By "FEM software" I mean "finite element METHOD software" -- not "finite element modeling software". FEM software in this context includes modeling, analysis and results display.

There are basically three different types of FEM software:

- Type 1: Bundled Modeling and Analysis. This type of software, usually from an established FEA vendor, has as its center an analysis module. Geometry creation and meshing capabilities, to the extent they exist, have been added as a front end. Results display capabilities are also usually present. MSC/pal 2 is an example of this type of software.

- Type 2: Bundled CAD and Modeling. This type of software has as its center CAD technology, to which has been added finite element modeling and results display capabilities. Computervision's CADDS FEM is an example of this catagory.

- Type 3: Unbundled Modeling. This type of software has as its center the capabilities of geometry creation, finite element modeling, and results display. Interfaces to analysis software and sometimes to CAD packages are available. An example of this type of software product is PATRAN.

Of course the above software categories are not absolute. There are plenty of hybrids in the market. For example, Computervision's Personal Designer Finite Element Modeler (PDFEM) is a hybrid between Type 2 and Type 3 software. It is tightly integrated with the Personal

Designer microCADDS software but it also can serve as a stand-alone modeler, with geometry construction capabilities of its own.

When looking at FEM software, it is important to assign a category to each product you look at. It is in this way that you'll begin to see which products are direct competitors and which products complement each other,can coexist in the same system, in your engineering office environment.

BEGINNER OR ADVANCED?

If you are a FEM beginner, you need to pay particular attention to product documentation, integration and support. You should ask what documentation is available for a particular product and perhaps ask to review copies of documentation prior to the sale. As a beginner, you should probably not consider buying any product if the documentation does not contain sections geared toward the beginner. Both conceptual material and a set of example problems are valuable.

As a beginner, you should also consider only that software which is tightly integrated. As a beginner, it can be difficult to ascertain how to achieve the desired end when two or three distinct products are at your disposal. You do not want to have to rely on general translators like IGES to transfer data between application codes.

INTERACTIVE GRAPHICS

Systems with interactive graphics provide several capabilities that aid the user in the finite element modeling task. The graphic representation of the geometry and finite element model is continuously on the graphics screen and updated to show the latest state of the model. You can identify parts of the model by interactively referencing existing parts of the model with a digitizing pen instead of typing in node or element numbers. For instance, a simple method to apply a displacement boundary condition to the model shown in Figure 1 is as follows. You could digitize each node location interactively, or even more simply, rotate the model so that the edge appears as a line, and identify all the nodes on the edge with a graphic window around the nodes affected. With interactive graphics you do not have to concern yourself with individual numeric tag values. To refine the mesh about a particular node, simply digitize that node as shown on Figure 2. You can also display with color what is usually considered nongraphic data. For instance, you can graphically display loads and boundary conditions, discriminate geometry by material or element properties.

GEOMETRY/MESH ASSOCIATIVITY

With a feature called geometry/mesh associativity, each piece of geometry (curve, surface or volume), is meshed individualy. The result is a group of elements which is associated to the original geometry. Element properties, such as plate thickness, or beam properties, and material descriptions are defined independent of the geometry. The element property sets and material descriptions are then assigned to the pieces of geometry. They are attached to the individual elements during the formatting process. If you want to change the element type or mesh density for a particular piece of geometry, you need only issue a new mesh command referencing the same geometry. The existing mesh is detected and automatically deleted prior to the new mesh. This makes for very flexible model generation. Distributed loads are applied to base geometry also, rather than individual elements. Again the loads are only attached to individual elements during the formatting process.

ADVANCED FEATURES

There are many advanced features usually found only in mainframe versions of finite element analysis codes that are now being made available on pc-based FEA codes. Some of these features include solid modeling, non-linear analysis, and substructuring. Typically finite element analysis requires large amounts of computer resources in terms of cpu and memory input/output speed, central memory, and disk storage. Within the limitations of the personal computer hardware, some FEA codes are implementing these more advanced features. For instance, although FEA analysis with a large number of solid elements can tax a mainframe, you might still want to build a course model with solid elements to run on a pc to get a rough idea of the behavior of a structure.

SUMMARY

Personal computers now have many of the same capabilities for finite element analysis that until recently were only available on much more costly hardware platforms. With this reduction in the entry cost, the technology is available to an increasingly larger group of users. Experience and education of this new group of users is of primary importance to insure proper usage of FEM methods. Novice users must rely on the same good judjment and common sense that they apply in other engineering areas to protect themselves from their own inexperience.

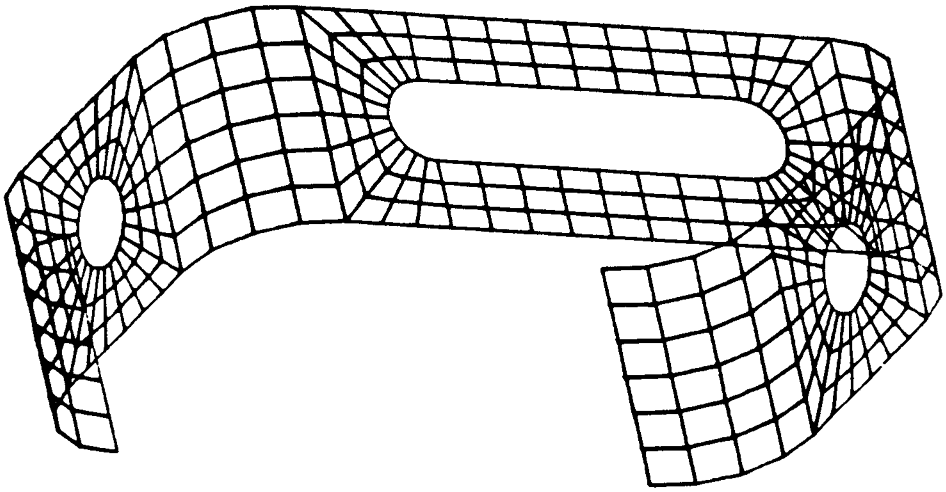

Figure 1a. Finite Element Model

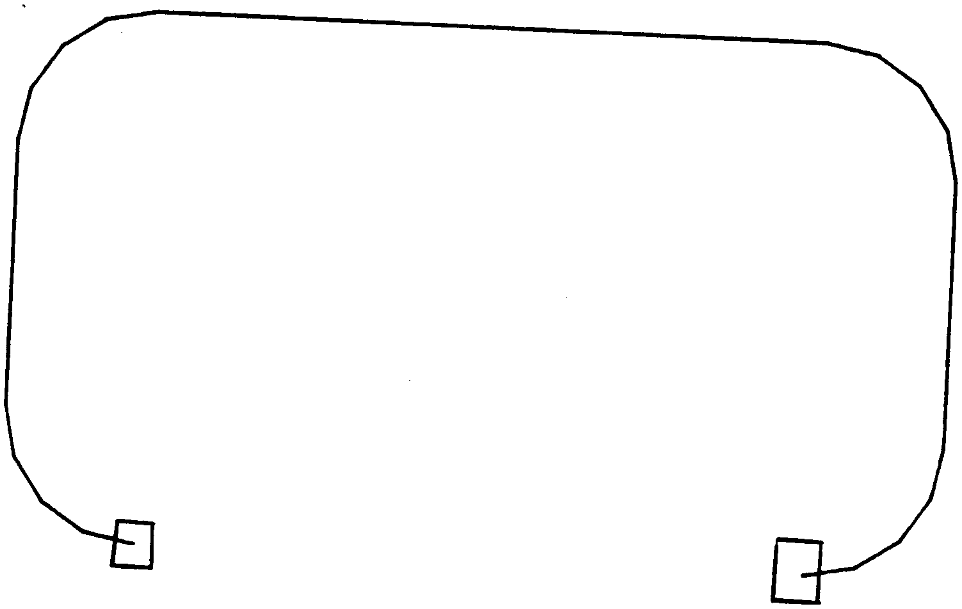

Figure 1b. Application of Displacement Boundary Conditions:
Insert cOnstr FIXED, X, Y, Z, RX, RY, RZ
Assign cOnstr FIXED; Win

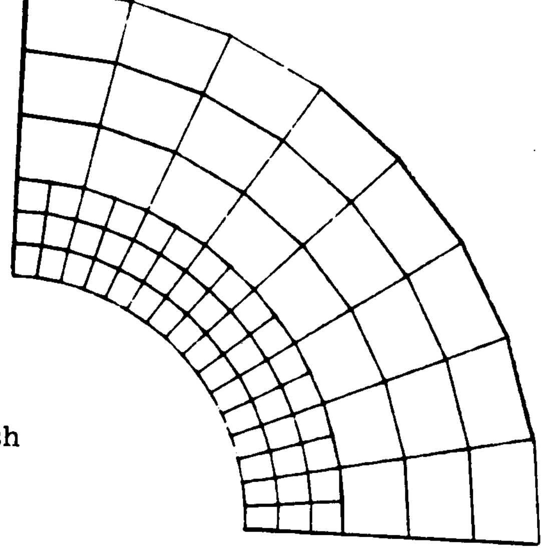

Figure 2a. Original Mesh

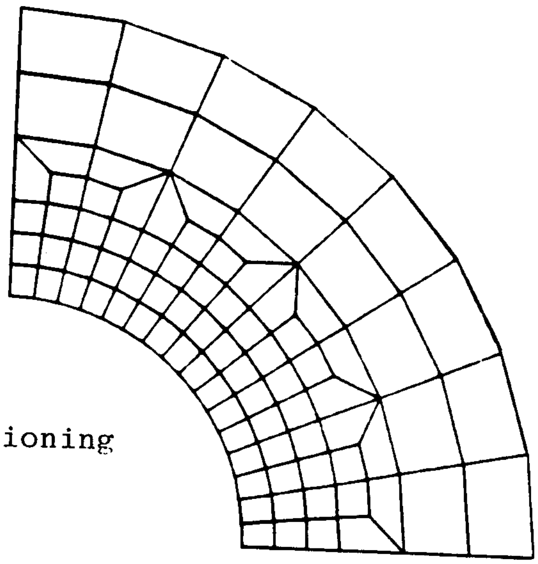

Figure 2b. Mesh Transitioning

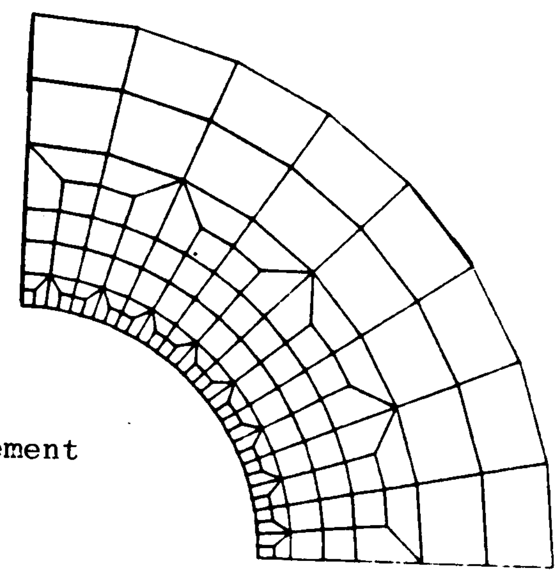

Figure 2c. Mesh Refinement