9 Finite Element Analysis Programs

9.1 Overview

Computer implementation of finite elements and solution procedures for engineering analysis is addressed. The end product is a general-purpose finite element analysis program. For such software to be used as an effective CAE tool, the programming should be hardware independent. The chosen finite elements and numerical methods must be accurate and reliable. The program should be executable on a given platform of choice—single processor, multi-processor, parallel processor, etc.

A general purpose FEA program consists of three modules: a pre-processor, a solver, and a post-processor. Commercial FEA programs can handle very large number of nodes and nodal degrees of freedom provided a powerful hardware is made available. User’s manual, theoretical manual, and verification problems manual, document a commercial FEA program.

Surveys of general-purpose programs for finite element analysis have been published [9.1]. At present FEA programs are used rather than written. Understanding of the organization, capabilities, and limitations of commercial FEA programs is generally more important than an ability to develop or even modify a FEA code. The emphasis on programming the FEM which was a major preoccupation in many recent textbooks [9.2 to 9.4] is therefore absent in this book.

The purpose of this chapter is to describe the organization and desirable capabilities of a general-purpose FEA program. A brief description of widely distributed and extensively used commercial FEA codes is included so that the reader is aware of their current capabilities.

Benchmark constitutes a standard set of test problems devised to assess the performance of FEA codes.

The practical issue of developing a viable FEA program and its implementation in the PC environment is a much larger challenge. Typically, it involves hundreds of human year’s effort.

9.2 FEA Program: Organization

The four components shown in Fig. 9.1 are common to virtually all general-purpose FEA programs. The INPUT phase enables the user to provide information relating to geometric representation, finite element discretization, support conditions, applied loads, and material properties. The more sophisticated commercial FEM systems facilitate automated generation of nodes and elements and provide access to a material property database. Plotting of the finite element model is also possible so that errors, if any, in the input phase, may be detected and corrected prior to performing computations.

The finite element library comprises the element matrix generation modules. Herein resides the coded formulative process for the individual finite elements. Ideally, the element library is open-ended
9.3 FEA Program: Capabilities

Fig. 9.1 Components of a general purpose finite element analysis program

and capable of accommodating new elements to any degree of complexity. This phase generates the required element matrices and vectors.

The assembly module includes all matrix operations necessary to position the element matrices for connection to neighbouring elements and the connection process itself. The latter operation thereby produces the global matrix equation of the finite element model.

The solution phase operates on the governing matrix equation of the problem derived in the previous phase. In the case of a linear static analysis, this may mean no more than the solution of a set of linear algebraic equations for a known right-hand side. In the case of linear vibration and buckling analysis, this may mean the extraction of eigen values and eigen vectors. Transient response analysis will require computations over a time history of applied load.

Finally, the results phase provides the analyst with a record of the solution. The record is commonly a printed list of nodal d.o.f., element strains and stresses, reaction forces corresponding to constrained degrees of freedom and a host of other requested information. As in input phase, there is a trend toward graphical output of results such as plots of displacement and stress contours, modes of vibration and buckling, etc.

A commercial FEM system therefore consists of three basic modules: pre-processor; solver; and post-processor. These modules and their functions are illustrated in Fig. 9.2. The pre-processor allows the user to create geometry or input CAD geometry, and provides the tools for meshing the geometry. The solver takes the finite element model provided by the pre-processor and computes the required response. The post-processor takes the data from the solver and presents it in a form that the user can understand.

9.3 FEA Program: Capabilities

The desirable features of a general-purpose FEA program are a large number of material models; a good library of finite elements; a good number of analysis procedures; and ability to manage the associated data. A brief discussion on these follows.
Chapter 9: Finite Element Analysis Programs

9.3.1 Material models

To cover a large number of metallic and non-metallic materials and a wide range of their behaviour, a general-purpose FEA program should provide a library of material models.

- Homogeneous, isotropic, linear, elastic
- Orthotropic
- Anisotropic
- Laminated composite
- Nonlinear elastic
- Elastic plastic
• Viscoelastic
• Viscoplastic
• Hyperelastic
• Temperature-dependent material properties

9.3.2 Element library

The available elements are for solid, structural, thermal and fluid flow analysis. They can be classified as follows:

• One-dimensional elements
  – 1-D, 2-D, 3-D bar elements
  – Linear/quadratic/cubic in order

• Two-dimensional elements
  – Triangular/quadrilateral in shape
  – Linear/quadratic/cubic in order
  – With straight/curved edges

• Axisymmetric ring elements
  – Triangular/quadrilateral in shape
  – Linear/quadratic/cubic in order
  – With flat/curved surfaces

• Three-dimensional elements
  – Tetrahedra/hexahedra/pentahedra in shape
  – Linear/quadratic/cubic in order
  – With flat/curved faces

• Beam elements
  – Euler–Bernoulli theory/shear deformation theory
  – 1-D, 2-D, 3-D beam elements

• Plate elements
  – Kirchhoff theory/Mindlin theory
  – Triangular/quadrilateral shapes
  – Linear/quadratic/cubic in order
  – With straight/curved edges

• Shell elements
  – Flat shell elements/facet approximation
  – Curved shell elements: triangular/quadrilateral shapes; quadratic/cubic orders
  – Axisymmetric shell elements: with curved surfaces; linear/quadratic/cubic in order
Chapter 9: Finite Element Analysis Programs

- Special elements
  - Spring
  - Gap
  - Rigid link
  - Contact
- Crack tip elements

Some of these elements are formulated to handle large displacements, large rotations and finite strains. Some formulations use reduced integration with hourglass control.

9.3.3 Procedures library

- Linear static analysis
- Linear dynamic analysis
  - Free vibration
  - Forced vibration
  - Transient response: mode superposition
  - Transient response: direct integration
  - Acoustic excitation and response
  - Spectrum response
- Linear buckling analysis
- Non linear analysis
  - Geometric nonlinearity
  - Material nonlinearity
  - Combined geometric and material nonlinearity
  - Contact problems
- Aero-elastic analysis
  - Divergence
  - Flutter
- Design optimization (sensitivity analysis)
- Thermal analysis: computational
- Fluid dynamics: computational
- Fracture mechanics: computational
- Electromagnetics
- Electromagnetics
- Electromagnetics
- Electromagnetics
- Magnetostatics

This allows the user to perform a wide variety of analyses. These procedures provide solutions for linear or nonlinear behaviour under static or dynamic loads. Large deformation and finite strain problems, contact problems, can also be addressed using these procedures.
9.3.4 Data processing

- Super elements
- Automated multilevel sub structuring
- Fourier analysis: axisymmetric bodies/shells under non-axisymmetric loads
- Cyclic symmetry
- Efficient numerical methods
  - Direct solver
  - Iterative solver
- Efficient computer systems
  - Super computers
  - Parallel processing systems
- Automatic adaptive mesh refinement

9.4 FEA Program: A Catalogue

A brief description of widely distributed commercial FEA programs is included here so that the reader is aware of their current capabilities.

9.4.1 MSC.Nastran

NASA Structural Analysis (Nastran) is a general-purpose program based on the finite element method developed by MacNeal Schwendler Corporation (MSC). The associated pre- and post-processor is called MSC.Patran. This premier FEA software is now available on the PC and runs both on DOS and Windows operating systems.

MSC.Patran provides the industry’s most comprehensive and powerful tools for the creation of accurate finite element models. Backed by the world’s largest CAE support organization and enhanced by continual use at some of the largest manufacturers, MSC.Patran sets the standard for finite element pre- and post-processing.

MSC.Nastran is the world standard in finite element analysis solutions. Its analysis capabilities give the user the competitive edge. With open choice of platforms from desktop PCs to supercomputers, MSC.Nastran is available wherever it is needed. MSC.Nastran’s unique element technologies provide highly accurate results with lower modelling effort, less solution time, and reduced computer requirements. Using MSC.Nastran one can optimize designs without increasing design cycle time. MSC provides the best documentation, customer support, and user training.

Building better products lighter, stronger, safer, in less time, at less cost are the business benefits of FEA using Nastran.

Analysis procedures in MSC.Nastran include: structural statics; structural dynamics; heat transfer; aero-elastic; magnetic field; piezo electric; acoustic; and hydro-elastic.
Chapter 9: Finite Element Analysis Programs

9.4.2 NISA

Numerically Integrated Finite Elements for Systems Analysis (NISA) is a family of general-purpose finite element programs for PCs, workstations and supercomputers developed by Engineering Mechanics Research Corporation (EMRC). The associated pre- and post-processor is called DISPLAY. The distinguishing features of the NISA programs are: user-friendly documentation; excellent technical support; flexible purchase options; and best price/performance in the industry.

NISA offers independent modules for a variety of analysis: linear statics; nonlinear statics, dynamics; heat transfer; composites; optimization; fatigue and fracture; fluid dynamics; printed circuit boards; electromagnetic fields; kinematic and dynamic analysis of mechanical systems.

NISA provides an excellent library of isoparametric finite elements. A special module NISA.P ADAPT utilizes $P$ elements. This program continually increases the order of the polynomial on a fixed finite element mesh until a reasonable convergence is reached. $P$ refinement and properly designed mesh is efficient and reliable.

NISA offers interfaces to major CAD/CAM systems: pro/engineer; unigraphics; CATIA.

DISPLAY is a powerful interactive graphics pre- and post-processor, which makes complex finite element modelling and results interpretation a cinch.

These programs reflect the latest advances in CAE utilizing finite element methods.

9.4.3 MARC

The right answer for finite element analysis is the general-purpose program called MARC developed by MARC Analysis Research Corporation. Special features of this program are: fully integrated nonlinear solution; powerful automated 3-D contact; accurate, adaptive simulations; parallel processing; and multi-physics. The associated pre- and post-processor is called Mentat.

MARC and Mentat allow the user to perform a wide variety of structural, thermal, fluid, and coupled field analyses using finite element method. The analysis procedures provide solutions for simple to complex linear and nonlinear problems in engineering.

The capabilities in MARC include: linear; nonlinear; large deformation and finite strain; automated contact; and adaptive meshing.

MARC has an extensive library of metallic and nonmetallic material models: linear elastic; elastic plastic; elastomers; hyperelastic; rigid plastic flow; creep; viscoelastic; viscoplastic; poro-elasticity and soils; powder metallurgy; composites; and concrete.

Over 140 elements are available in MARC, which are modern, robust and accurate. They are grouped as: truss; beam; plane stress; plane strain; generalized plane strain; plate; shell; membrane; axisymmetric; 3-D solid; special elements (springs, gaps, rigid links, pipe bend, etc.); and user defined elements.

Analysis types supported by MARC are: statics; dynamics; heat transfer; thermo mechanical; fracture mechanics; fluid dynamics; hydrodynamic bearing; joule heating; acoustics; electrostatics; magnetostatics; electromagnetics; design sensitivity and optimization.

Mentat is tightly integrated with the MARC FEA program, allowing all data to be defined interactively through a powerful graphical user interface. Notable capabilities include: geometry creation; solid modelling; mesh generation; analysis support; post-processing; and advanced rendering.

Mentat’s optional modules support interfaces to the leading CAD/CAM systems CATIA; pro/engineers; I-DEAS, and auto CAD.
9.4.4 LS-DYNA

LS-DYNA is a general-purpose code based on the FEM for analyzing large/elastic/inelastic deformation dynamic response of solids and structures including structures coupled to fluids. The main solution procedure is based on explicit time integration. An implicit solver is also available with somewhat limited capabilities for structural and heat transfer analysis.

A contact impact algorithm allows difficult contact problems to be easily treated with heat transfer included across the contact interfaces.

Spatial discretization is achieved by the use of four-node tetrahedral, eight-node hexahedral solid elements; two-node beam elements; three-node triangular and four-node quadrilateral shell elements; eight-noded solid shell elements; truss elements; membrane elements; discrete elements; and rigid bodies. A variety of formulations are available for each element type (solid, fluid, structural, discrete).

Specialized capabilities for modelling airbags, sensors, and seat belts have tailored LS-DYNA for applications in the automotive industry.

Adaptive meshing is available for shell elements and is widely used in sheet metal stamping simulations.

LS-DYNA currently has over two hundred material models and over ten equations of state to cover a wide range of material behavior.

LS-DYNA is operational on supercomputers, mainframes, workstations, parallel processing systems, and PCs.

The associated pre- and post-processor is called LS-TAURUS.

LS-DYNA and LS-TAURUS are developed by Livermore Software Technology Corporation.

9.4.5 ANSYS

ANSYS is an integrated design analysis tool based on the FEM developed by ANSYS, Inc. It has its own tightly integrated pre- and post-processor. The ANSYS product documentation is excellent and it includes commands reference; operations guide; modeling and meshing guide; basic analysis procedures guide; advanced analysis guide; element reference; theory reference; structural analysis guide; thermal analysis guide; electromagnetic fields analysis guide; fluid dynamics guide; and coupled field analysis guide. Taken together, these manuals provide descriptions of the procedures, commands, elements, and theoretical details needed to use the ANSYS program. All of the above manuals except the ANSYS theory reference are available online through the ANSYS help system, which can be accessed either as a standalone system or from within the ANSYS program. A brief description of the information found in each of the manuals follows.

Engineering capabilities of ANSYS products are: structural analysis (linear stress, nonlinear stress, dynamic, buckling); thermal analysis (steady state, transient, conduction, convection, radiation, and phase change); CFD analysis (steady state, transient, incompressible, compressible, laminar, turbulent); electromagnetic fields analysis (magnetostatics, electrostatics); field and coupled field analysis (acoustics, fluid–structural, fluid–thermal, magnetic–fluid, magnetic–structural, magnetic–thermal, piezoelectric, thermal–electric, thermal–structural, electric–magnetic); sub-modelling; optimization; and parametric design language.

Element library in ANSYS lists 189 finite elements. They are broadly grouped into: LINK, PLANE, BEAM, SOLID, CONTAC, COMBIN, PIPE, MASS, SHELL, FLUID, SOURCE, MATRIX, HYPER, VISCO, INFIN, INTER, SURF, etc. Under each type, different shapes and orders complete the list. Obviously, ANSYS has the best elements in its library.
Chapter 9: Finite Element Analysis Programs

Analysis procedures in ANSYS can be grouped into: static analysis, transient analysis, mode frequency analysis, harmonic response analysis, buckling analysis, sub-structuring analysis, and spectrum analysis.

In ANSYS, there are two fundamentally different types of optimization. The first is referred to as design optimization; it works entirely with the ANSYS parametric design language and is contained within its own module (ANSYS /OPT). The second is topology optimization, a form of shape optimization.

ANSYS finite element analysis software enables engineers to perform the following tasks:

- Build computer models or transfer CAD models of structures, products, components, or systems.
- Apply operating loads or other design performance conditions.
- Study physical response, such as stress levels, temperature distributions or electromagnetic fields.
- Optimize a design early in the product development process to reduce production costs.
- Do prototype testing in environments where it otherwise would be undesirable or impossible.

9.5 Closure

Spectacular advances have been made in the development, documentation, and implementation of commercial FEA programs on PCs, workstations, mainframes, and supercomputer systems. Pre-processors with graphical user interface are also available that can create finite element models of virtually all CAD models. Post-processors are capable of display and animation of the results of every finite element analysis. At present, FEA programs have been integrated in widely used CAD/CAM systems. Computer implementation of finite element procedures is not trivial; it involves hundreds of human years effort not only for development but also for updates.

It is instructive to compare and contrast the desirable features of a general purpose FEA program with the current capabilities of commercial FEA codes. This may provide directions for modifications, extensions and upgrading of commercial FEA codes.

It is recommended that the reader use one of the commercial FEA programs, not necessarily from those described here, to analyze the computational problems listed in the text. This will enable the user to acquire the skills needed to effectively use the FEM in general, and a general-purpose program in particular, in practice.

Advanced applications of the FEM, not considered so far, can be attempted using commercial FEA programs. Some of these are identified and described in the next chapter.

9.6 References