"ANALYSIS MODELING" with FAM

by

Joseph L. Walsh
General Manager, US Operations
FEGS Limited
Duluth, GA 30136

ABSTRACT

FEGS Limited introduces a new approach of modeling for Finite Element Analysis applications based on an "intelligent" CAD interface and full Geometric Associativity of analysis data.

This paper describes and compares the processes related to Finite Element Pre/Post Processing, Design Modeling, and " Analysis Modeling " .

This paper also discusses the CAD interface technology and Geometric Associativity provided by the FAM (Field Analysis Modeler) software available from FEGS Limited.

Introduction

The past ten (10) years have seen the role of Finite Element applications evolve from that of failure analysis through component/product verification to component/product design. The migration of Finite Element applications up the the design process by most organizations has resulted in the development of a Design/Analysis Cycle consisting of several analyses of design alternatives and revisions to design being driven by analysis results, requiring cumbersome recreation of Finite Element models of design alternatives and revisions. This migration has also required tighter integration of CAD technology and Finite Element technology.

Design/Analysis Models

A detailed review of the Design/Analysis Cycle requires a detailed description of the models used throughout the process. The Design/Analysis Cycle typically involves three (3) major models representing the product/component . The first model is the geometry based Design Model and typically resides in a CAD system. The second model is a geometric Idealized Model for the purposes of anaylsis. The third model is a mesh based Analysis Model to be analyzed.

The purpose of the Design Model is to model geometry of the component/product at various stages of design. This geometry model may be a solid model, a surface model, a wireframe model or 2-Dimensional drawings. The Design Model is typically very accurately detailed and readily changed.

The Idealized Model is a secondary geometry model created from the original Design Model through a process of Model Abstraction. The purpose of the Idealized model is to represent the idealized geometry of the component/product for analysis purposes. This geometry model usually varies considerably in form from the Design Model to take into account such items as symmetry, cyclic symmetry, detail removal, dimensional reduction (ie. shell/plate models), contact interfaces, and many others. The Idealized Model is usually created in the Finite Element Pre-Processing system.

The Analysis Model is created for the purpose of predicting component/product behavior. This model consists of a Finite Element mesh, loads, properties and boundary conditions for analysis. The Finite Element mesh created is a function of the Idealized Model geometry, therefore, since the Analysis Model is intended to model behavior and not geometry, the Idealized Model typically consists of a geometry definition that is different than that of the Design Model. The Analysis Model is an approximate discretized representation and is typically difficult to modify. The Analysis Model is analyzed using MSC/NASTRAN in order to obtain obtain results of behavior which can be evaluated by the analyst.

Design/Analysis Cycle

The Design/Analysis Cycle involves an iterative process whereby several design revisions and/or alternatives are evaluated based on results from MSC/NASTRAN analyses. This iterative process is illustrated in Figure 1 below:

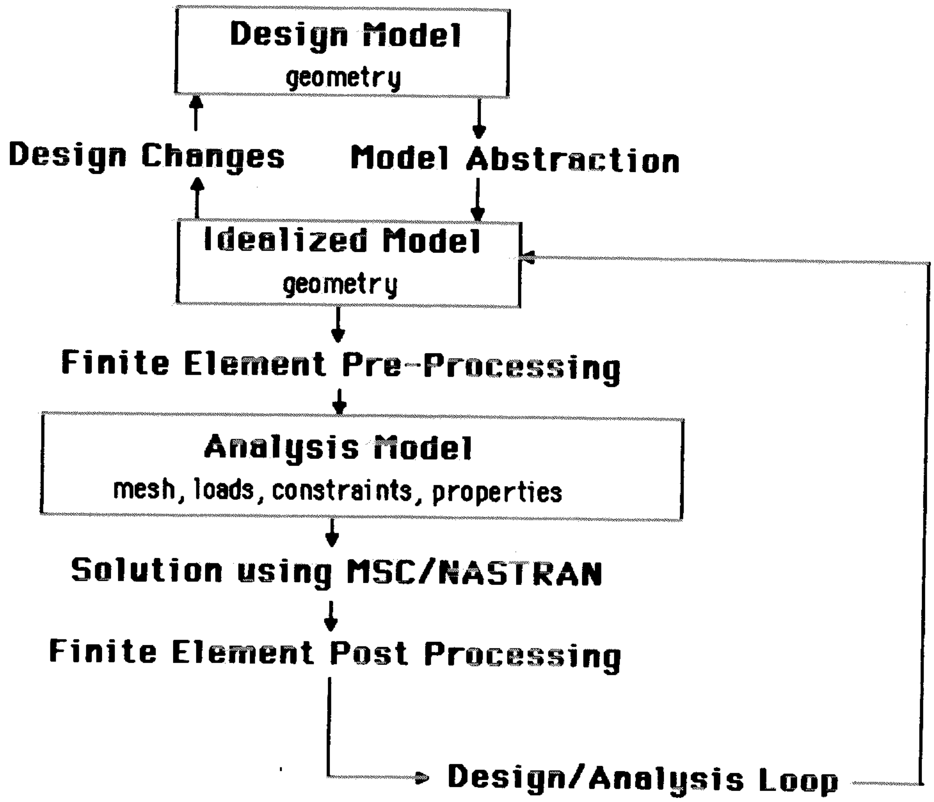

Figure 1: Design/Analysis Cycle

The Design Model is generated in the CAD system representing the geometry of the component/product under consideration. This model may be a solid model, surface model, wireframe model , 2-Dimensional drawings, or a combination of any of these forms.

The Design Model is then transferred from the CAD system to the Finite Element Pre-Processing system and converted by the analyst to an Idealized Model representing the geometry to be analyzed. This process of converting the Design Model to the Idealized Model is referred to as Model Abstraction. The Idealized Model is a geometric representation which will allow for prediction of a desired behavior of the component/product when analyzed with MSC/NASTRAN. Detail removal and dimensional reduction are common model abstractions.

The Idealized Model is then meshed using either mapped meshing or free meshing algorithms and loads, constraints, properties, and analysis directives are added to form a complete Analysis Model for solution. This Analysis Model is then converted to an MSC/NASTRAN data file ready for analysis.

The Analysis Model is analyzed using MSC/NASTRAN for solution. The results of the analysis solution are then read into a Finite Element Post-Processing system allowing the analyst to graphically review the behavior of the analysis model.

Review of the Analysis Model behavior versus desired objectives causes the analyst to incorporate design changes into his Idealized Model in order to prepare a new Analysis Model. The new Analysis Model is then analyzed and behavior reviewed by the Analyst. This process of looping between the Idealized Model and the Analysis Model is referred to as the Design/Analysis Loop and is repeated until desired behavior is achieved or design time constraints are encountered. Completion of the Design/Analysis Loop results in design recommendations being forwarded back to the Design Model.

Design/Analysis Methodologies

The Design/Analysis Cycle has been described, without reference to the actual methodologies used, to acheive this cycle and their resultant process flow, advantages and disadvantages. There are currently three (3) major methodologies provided by today's software technology for providing tools for the Design/Analysis Cycle. The first methodology of interest is that of Finite Element Pre/Post-Processing, the second methodology under consideration is referred to in this paper as Design Modeling and involves the use of a core Design Model, the third methodolgy available is that of " Analysis Modeling " based on the FAM software available from FEGS Limited.

Finite Element Pre/Post-Processing

This is the most commonly used process for Design/Analysis applications and has been available for several years. A process flow diagram of Finite Element Pre/Post-Processing is illustrated in Figure 2 of this paper.

The Design Model is transferrred to the Finite Element Pre-Processing system via IGES typically in a wireframe format often losing detail essential to the Idealized Model or carrying over excessive detail such that the analyst often prefers to create the Idealized Model from scratch. The Model Abstraction process is a totally manual process carried out by the analyst based on experience.

The analyst then uses sophisticated meshing algorithms to generate a Finite Element mesh based on the Idealized Model. Loads, boundary conditions, constraints, and properties are added to the Finite Element mesh to form a complete Analysis Model. This Analysis Model is then analyzed using MSC/NASTRAN and results are reviewed graphically by the analyst.

This methodology does not provide any built-in feedback loop and often requires the analyst to create a new Idealized model from scratch for design changes or at a minimum forces the analyst to redirect the meshing algorithms and respecify loads, constraints and properties on the mesh.

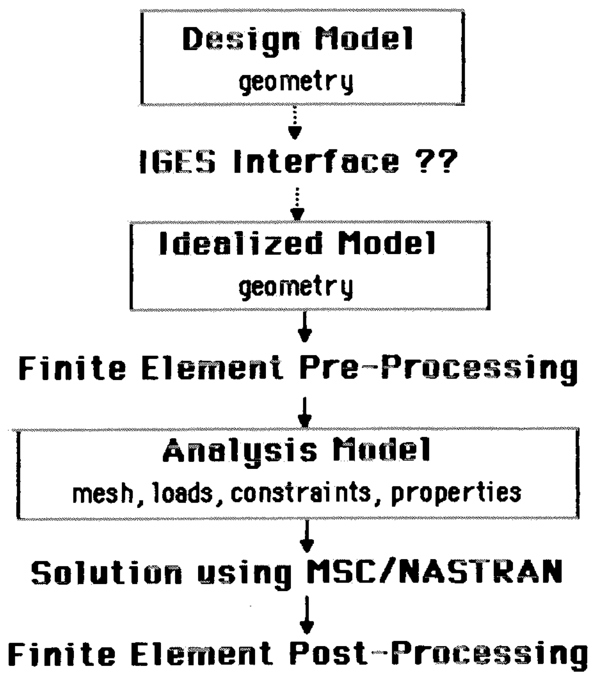

Figure 2: Finite Element Pre/Post Processing

The advantages of the Finite Element Pre/Post Processing Methodology are as follows: it provides for Model Abstraction between Design Model and Idealized Model, it provides excellent user control of mesh flow and quality, it allows large quantities of output to be comprehended at a glance, and Finite Element Pre/Post-Processing is a proven methodology.

The disadvantages of the Finite Element Pre/Post Processing Methodology are : a poor interface from a Design Model resulting in cumbersome regeneration of geometry for an Idealized Model, no feedback loop available for design changes, design changes require respecification of mesh level attributes and properties, Design/Analysis iterations are difficult and time consuming.

Design Modeling

Design Modeling in this paper refers to a methodology for Design/Analysis applications which is based on a core Design Model, typically a solid model, and automatic free meshing algorithms to generate a Finite Element mesh. This methodology has become availailable within the last few years and is is illustrated in figure 3 below.

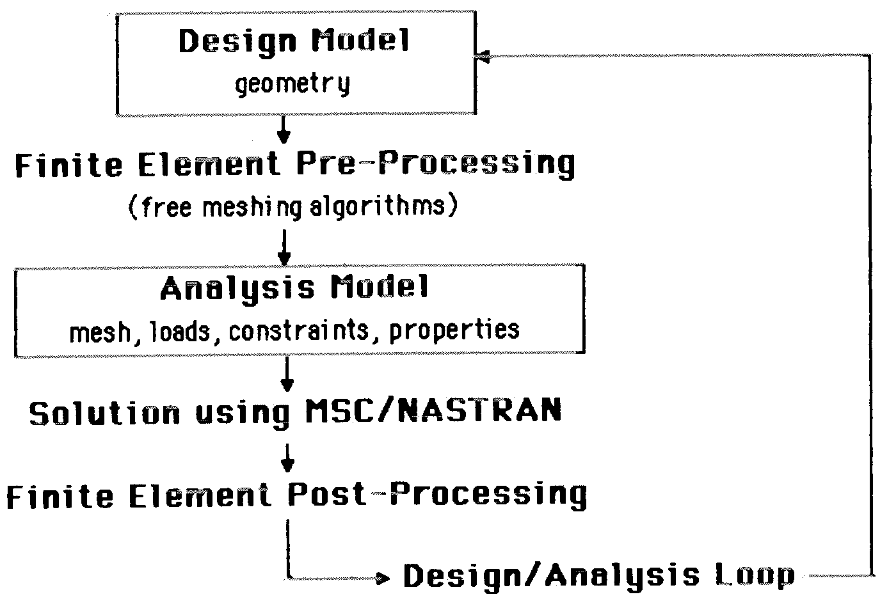

Figure 3: Design Modeling

The Design Model is generated by the user as a base for all modeling applications. The Design Model is directly converted to an Analysis Model through free meshing algorithms and loads, constraints, etc. are added by the user. There is typically no Idealized Model in this approach, therefore, the user must either analyze the geometry of the Design Model or utilize parrallel dissaciocated Design Models (one for analysis and one for design). If the user selects parrallel models for different purposes then he is actually using the Finite Element Pre/Post-Processing Methodology described earlier.

The Analysis is then submitted for analysis using MSC/NASTRAN and results are viewed graphically by the user on the Design Model. Design changes are made directly to the Design Model and the process is repeated until the desired design objectives are achieved. Changes are therefore made to the Design Model prior to any knowledge of the effective change in behavior due to the design change.

Some feature based systems allow supression of features when going from the Design Model to the Analysis Model thereby providing some Model Abstraction. However, the supression of features between the Design Model and the Analysis Model must be be repeated at each step of the Design/Analysis Loop.

The advantages of the Design Modeling methodology are as follows: no interface required from the Design Model, it provides a feedback loop with changes made directly to the Design Model, and it provides "free" meshing algorithms to simplify creation of the Analysis Model.

The disadvantages of the Design Modeling methodology are as follows: it requires analysis of the Design Model geometry or repeated feature suppression at each iteration, it utilizes limited element types (ie. tetrahedrons only for solids), it is difficult to control mesh flow and quality, it produces large models for analysis as compared to Finite Element Pre/Post-Processing, and all design revisions are reflected in all aspects of design prior to verification of performance.

" Analysis Modeling "

A third methodology is currently available for Design/Analysis applications based on the FAM suite of software provided by FEGS Limited. This Methodology is referred to in this paper as " Analysis Modeling " and has been designed with the Design/Analysis Cycle in mind. The process of " Analysis Modeling " utilizes an "object-oriented" entity-based database with full Geometric Associativity between the Idealized Model and the Analysis Model. This Geometric Associativity is coupled with an intelligent CAD interface allowing for Model Abstraction in order to provide a complete system for " Analysis Modeling ".

The Design Model is transferred to the Idealized Model using the IGES format and the FAMfromCAD software module. FAMfromCAD provides the analyst with a Model Abstraction tool to cleanup IGES geometry, perform automatic detail removal, perform automatic generation of higher level geometry (ie. surfaces, solids) from wireframe geometry, and automatically distribute mesh control parameters on the Idealized Model.

Mesh control parameters are specified by the analyst on the geometric Idealized Model. Loads, boundary conditions and properties are also assigned to the geometric entities of the Idealized Model. The FAM software then automatically generates a mesh based on the mesh control parameters using mapped meshing algorithms retaining associativity between the idealized Model geometry and the Analysis Model mesh. The loads, boundary conditions, and properties are then distributed to the appropiate mesh entities associated with the specified geometric entities.

The Analysis Model is then submitted to MSC/NASTRAN for solution and the results are read back into the FAM software allowing graphical display of behavior on either the Analysis Model mesh entities or the Idealized Model geometric entities.

Design revisions are then made to the Idealized Model geometry by the analyst. These changes are then transferred automatically to a new revised Analysis Model based on the Geomtric Associativity discussed previously. All revisions to the Idealized Model geometry are automatically incorporated into a new Analysis Model by the FAM software providing a Design/Analysis Loop capability. The process is then repeated until the desired behavior is achieved at which point the Idealized Model geometry is used to specify recommended design changes to the Design Model as illustrated in Figure 4 below.

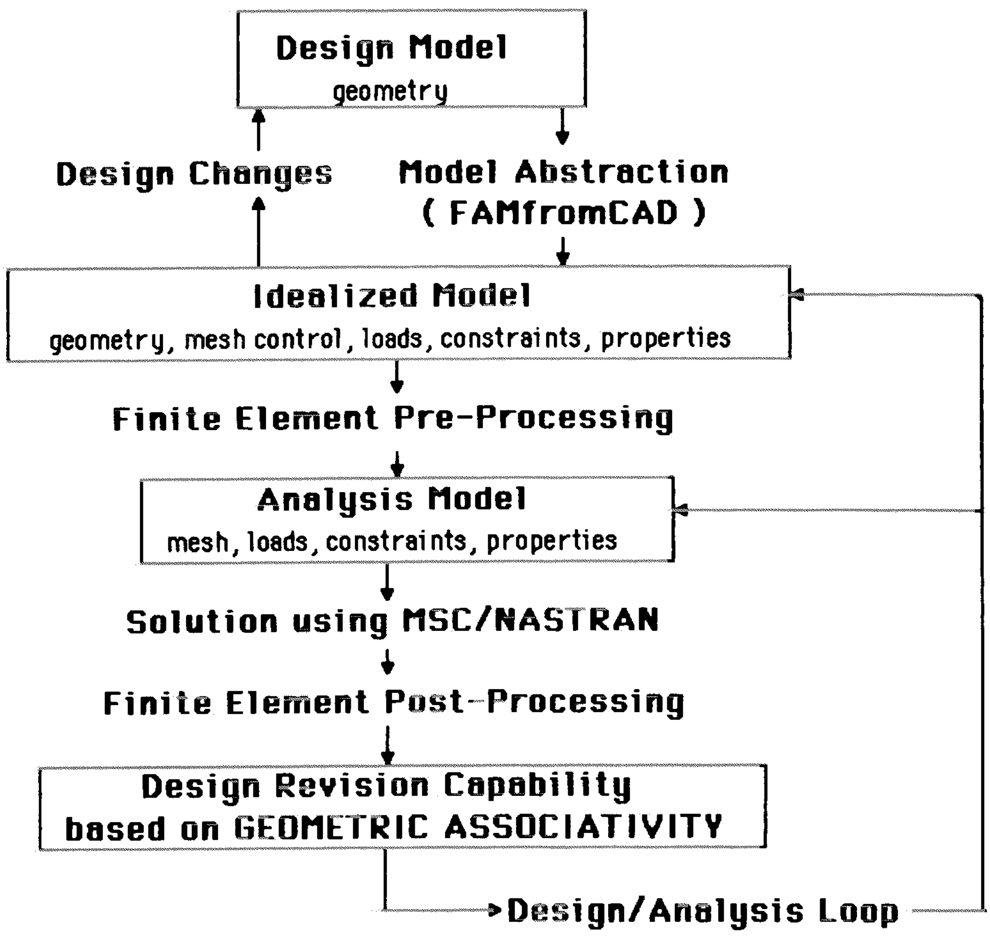

Figure 4: " Analysis Modeling "

The major advantages of the " Analysis Modeling" methodology are as follows: an intelligent interface with the Design Model (FAMfromCAD) provides a tool for Model Abstraction, the geometric entity based modeling allows for working at concept level, Geometric Associativity between the Idealized Model and the Analysis Model provides an automatic Design/Analysis Loop, and it provides excellent control of mesh flow, quality, and element types.

The major disadvantage of the " Analysis Modeling " methodology is that it is a new methodology which requires retraining and time for the analyst to become proficient.

FAMfromCAD "intelligent" CAD Interface

The transfer of geometry data from the Design Model to the Idealized Model using the IGES format can lead to a wide variety of difficulties in obtaining the desired geometry in the Idealized Model. Some of the common difficulties encountered include excessive detail transfer, disconnected geometry, duplicate points and lines, and generally a reduction of the Design Model to a poorly connected 3-Dimensional wireframe model. The geometry data transferred through IGES does not reflect any Model Abstraction, unless this was performed in the CAD system prior to transfer, and usually requires major modification for the purpose of Model Abstraction in the Finite Element Pre-Processing system.

FAMfromCAD attempts to resolve some of these difficulties by providing a hierarchical ladder of commands with "intelligent" direction taken from the user by way of user-defined control parameters. The geometry is processed based on these control parameters and a new geometry for the Idealized Model represented in the FAM software. This processing allows FAMfromCAD to build Solid Models from poorly connected wire frame data. Figure 5 below illustrates the the geometry processing ladder used by FAMfromCAD.

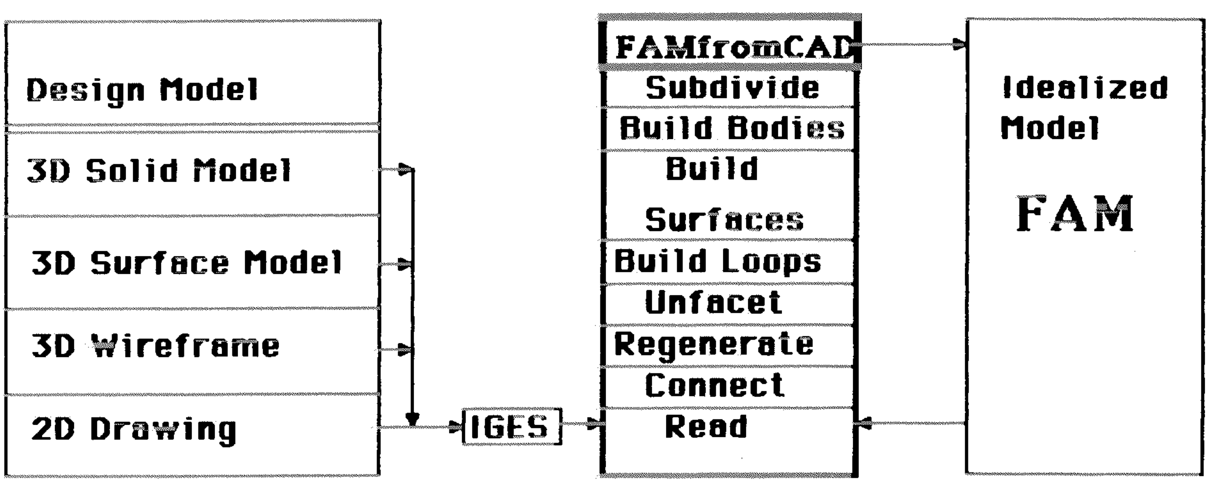

Figure 5: FAMfromCAD Geometry Processing

The geometry is transferred from the Design Model to IGES. FAMfromCAD then reads this IGES file and processes the geometry to generate the initial geometry for an Idealized Model in FAM. This geometry processing provides the user with a tool to assist in Model Abstraction. The Idealized Model geometry is then read by FAM, at any stage of geometry processing, and is then available for manual manipulation by the analyst. The geometry may then be read again by FAMfromCAD and reprocessed iteratively until the desired Idealized Model geometry is achieved. This iterative processing of geometry data by FAMfromCAD provides a semi-automatic method guided by the user for Model Abstraction from a Design Model to an Idealized Model.

FAMfromCAD Geometry Processing

The first step in the geometry processing phase is to read the IGES file geometry data. This file often includes data which is irrelavent to the analyst (ie dimensions, annotations, etc.). FAMfromCAD filters out any unnecessary data during the Read process. The geometry at this stage is typically a poorly connected wireframe representation of points, lines, and arcs in 3D space. Figure 6 below illustrates a sample geometry read from IGES.

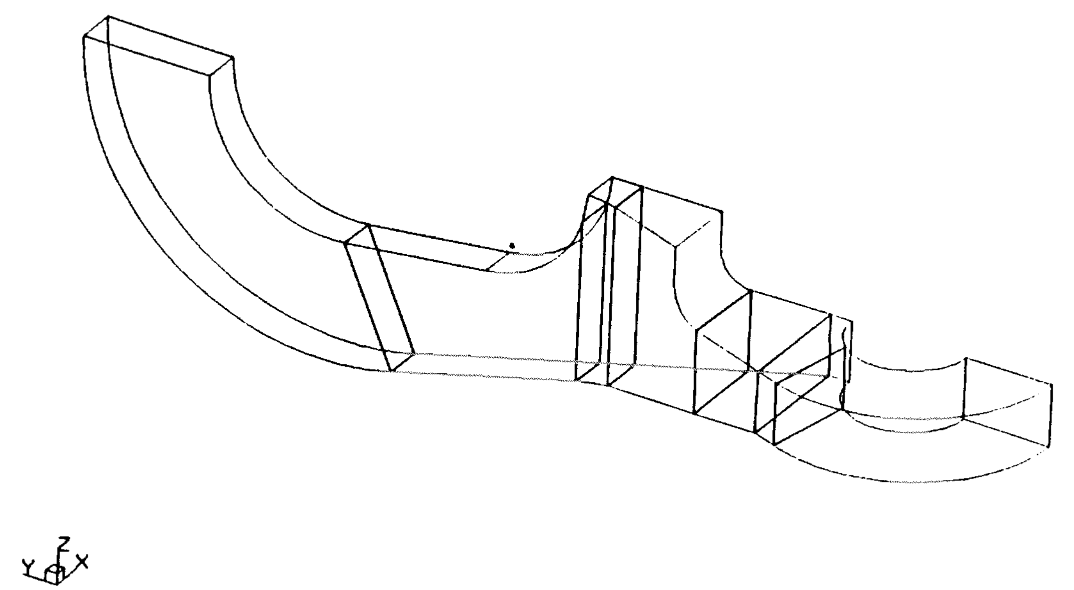

Figure 6: Geometry Read from IGES File

The geometry is then processed through the Connect, Regenerate, and Unfacet facilities which will clean up ambiguous geometry and disconnected geometry as well as unfacet short lines into long straight lines, arcs, or splines as appropriate based on user defined control parameters for geometric tolerance and minimum facet lengths. The geometric tolerance may be set for a different value at each stage of geometry cleanup or remain constant. The Connect facility collapses any duplicate geometry within geometric tolerance, thereby providing the user with a tool for detail removal and dimensional reduction. The geometry at this point may vary from the Design Model and is a well connected wireframe representation of the component/product.

The geometry is then run through the topology building facilities of FAMfromCAD. These topology building facilities consist of: Build Loops which scans the entire model for loops of closed lines in 3D space and stores them as potential surfaces, Build Surfaces which scans the loops of potential surfaces and creates all surfaces which would be valid based on control parameters specified by the analyst, and Build Bodies which scans all the surfaces in the model and finds any closed loops of surfaces which could be used to form a body definition and creates bodies whenever a valid definition can be found. Figure 7 below illustrates the solid bodies found by FAMfromCAD for the sample geometry illustrated in Figure 6. The completeness and accuracy of the Idealized Model generated by FAMfromCAD is a function of the quality of data first received from IGES, the user specified control parameters, and the use of FAM for manual guidance at intermittent stages.

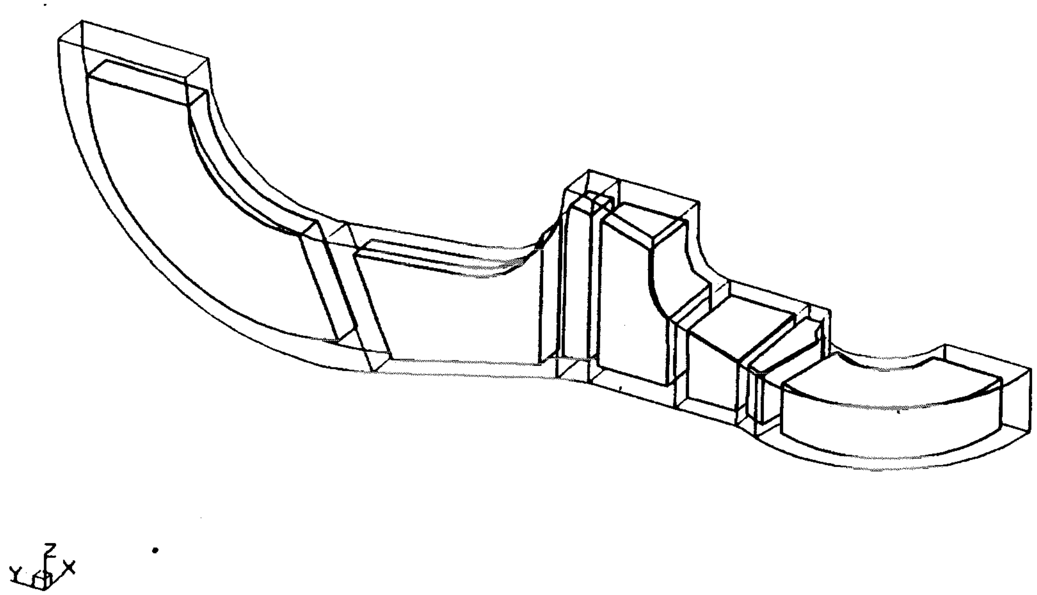

Figure 7: 3 D Solid Geometry generated by FAMfromCAD

The geometry processing of FAMfromCAD is completed with the Subdivide facility which distributes the mesh control parameters to the geometry based on user-defined control parameters for maximum element length and minimum number of divisions. This facility gives the user the feel of "free" meshing while maintaining all of the control inherent in a mapped meshing system.

FAM Geometric Entity Based Database

The Idealized Model geometry existing in the FAM database (whether received from FAMfromCAD or generated from scratch) has unique properties of full geometric connectivity and geometric hierarchy. These properties are provided by FAM's entity based ("object oriented") database structure and the hierarchy built into that structure. The FAM geometric database consists of five (5) geometric entities as illustrated in Figure 8.

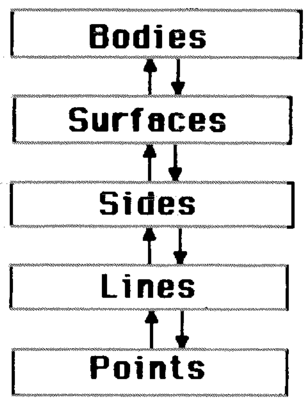

Figure 8: FAM Geometric Entities

Points are defined in FAM as locations in X,Y,Z space. Lines may be straight lines, arcs, splines, or intersection curves and are fully defined by the points which make up the lines. The definition of the lines in the database contains only the points used to create the line and does not store any X,Y,Z coordinates of the points with the line definition. Sides consist of one (1) to four (4) lines to be referenced in a surface definition. Surfaces are defined by three (3) or Four (4) sides which bound the surface. This concept of Sides and Surfaces allows FAM to create compatible surface definitions for the geometry in Figure 9. Bodies are defined by Surfaces which bound the Bodies in 3D space.

Any geometric entity or group of geometric entities may be placed, referenced by name, in an entity called a Set. This Set mechanism allows for groups of geometric entities to be manipualted as a single entity (ie. the top face of a component) and any operation available for a geometric entity is also available for a Set.

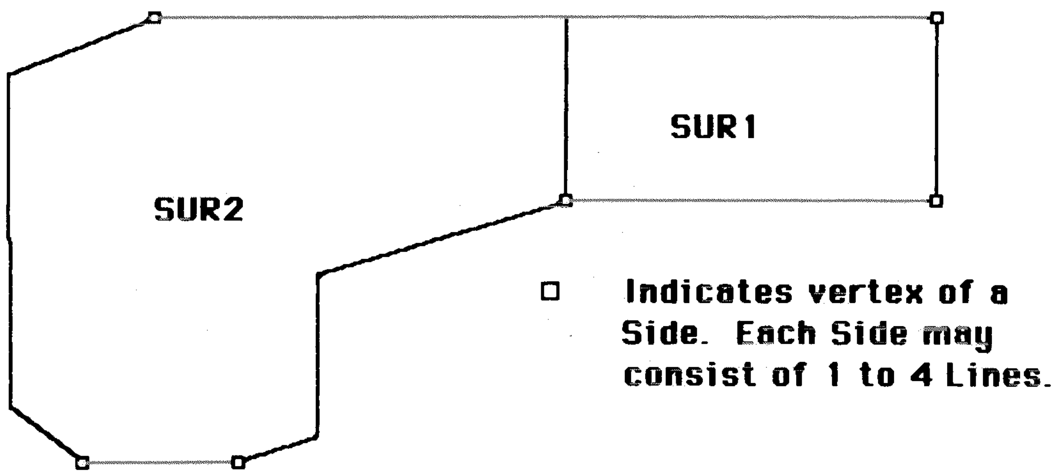

Figure 9: FAM Surface/Side/Line relationship

The FAM data structure allows for full two (2)-way hierarchy of the geometric entities and maintains full geometric connectivity of the database. The geometry may be modified at any level of geometric entity (Bodies, Surfaces, Sides, Lines, or Points) and the modifications will automatically occur to all higher levels entities which reference the modified part and all lower level entities which are referenced by the modified part. This is possible since each geometric entity definition only contaions a reference to the lower level entities which make it up. Mesh control parameters are stored as attributes at the Line level thereby ensuring a consistent mesh across surfaces without compatible sides.

FAM Geometric Associativity

Geometric Associativity is one of the major characteristics of FAM and is acheived through the use of geometric entities and analysis entities. Loads, constraints, physical properties, and material proprties are stored as referencable entities in the FAM database and are then assigned as reference attributes to the geometric entities. This unique data structure allows these analysis entities to become an integral part of the geometric entity definitions and to remain assigned to the geometric entities during any modifications, it also allows any change to the analysis entities to be automatically distributed to the geometric entities by FAM.

Any change to geometry, mesh control, or analysis entities are reflected in all appropriate geometric entities. FAM then generates the mesh entities (grids and elements) automatically from the geometric entities and mesh control parameters, and distributes the analysis entities to grid and element loads, constraints, and properties by mapping the geometric entity attributes to the current mesh entities. This data structure allows for associativity of all mesh data and analysis data to geometry which is referred to as full Geometric Associativity.

FAM Design/Analysis Loop

The geometric hierarchy and connectivity inherent in the FAM data structure along with the Geometric Associativity of mesh dat and analysis data provide an excellent facility for design revision. An example of a complete Analysis Model including mesh entities and analysis entities generated in FAM is illustrated in Figure 10 and will be used to illustrate the process of design revision within FAM.

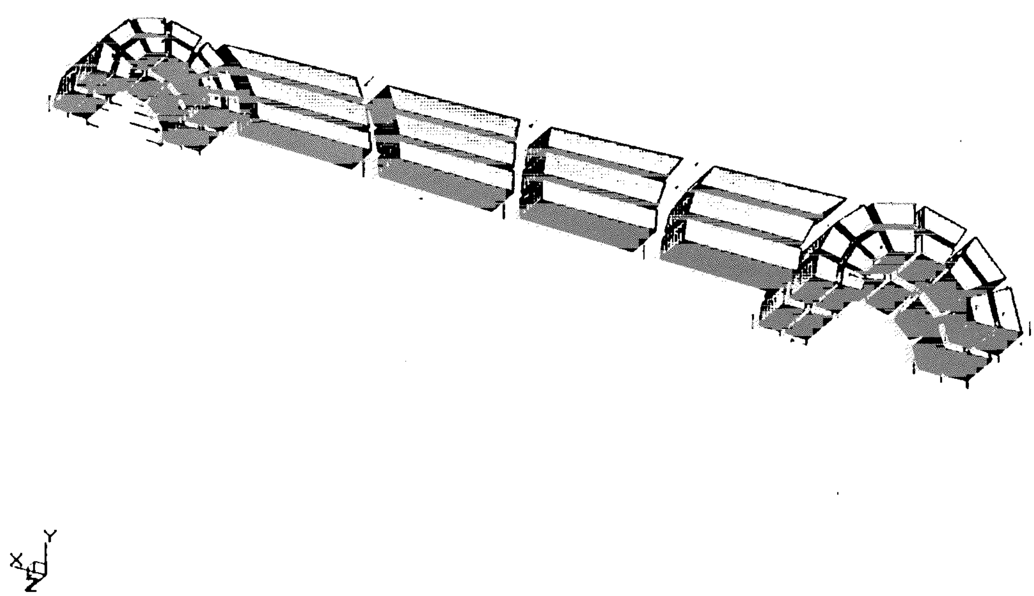

Figure 10: complete Analysis Model generated by FAM

The process of implementing a design revision in FAM is as follows:

1.) The user specifies a desired change to geometry and/or mesh control parameters as illustrated in Figure 11.

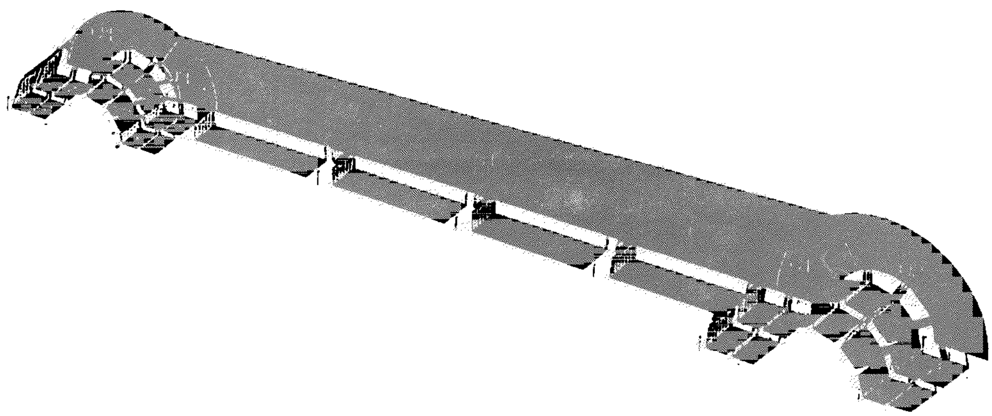

Figure 11: Design change specified by moving a set of Surfaces

2.) The user issues two (2) commands to FAM (MESH, LOAD)

3.) FAM automatically generates a new complete Analysis Model with appropriate mesh data and analysis data as illustrated in Figure 12.

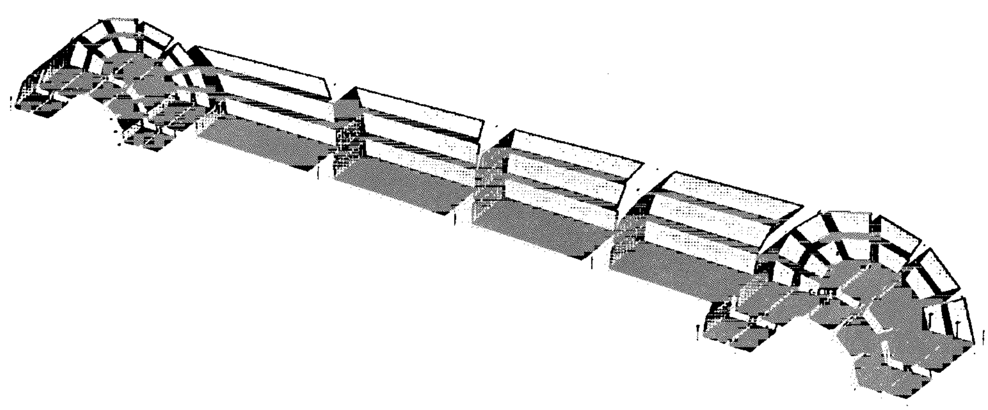

Figure 12: FAM automatically generates a new Analysis Model

Conclusion

"Analysis Modeling" with FAM introduces a new methodology for Design/Analysis applications based on an "intelligent" CAD interface and Geometric Associativity for the creation of MSC/NASTRAN Analysis Models. This new methodology has been designed specifically for the Design/Analysis Cycle and provides Model Abstraction tools and an automatic feedback loop to the analyst for design changes without sacrificing any of the benefits of the classical Finite Element Pre/Post-Processing approach.

"Analysis Modeling" may be beneficial to those organizations performing Design/Analysis functions who: 1) are having difficulty interfacing current CAD technology with MSC/NASTRAN, 2) are having difficulty incorporating design changes into their Finite Element models, or 3) would like to move their Finite Element applications further up the design process toward conceptual modeling.

References

G. Butlin and T. Ing, "Associative Analysis Modelling", October 1988, FEGS Limited, Oakington, Cambridge CB4 5BA, England

J. Aas, "A CAD-FEM Strategy", October 1988, FEGS Limited, Oakington, Cambridge CB4 5BA, England

J. Aas, "Associative Analysis Modelling and Optimisation", September 1988, FEGS Limited, Oakington, Cambridge CB4 5BA, England

F.G. Willis, "CAE in the Engineering Process of the 90's", May 1989, Ford Motor Company, Dearborn, MI

R.D. Lowrey, "A Comparison of Quadratic and Linear Hexahedrons and Tetrahedrons", May 1989, Automation Technology Products.

MSC/NASTRAN, The MacNeal-Schwendler Corporation