"Geometric" FE Analysis with MSC/XLplus

Celso A. Barcelos

Manager FEM/FEA

Aries Technology
600 Suffolk Street
Lowell, MA 01854

Abstract

Historically, finite element modeling has been done independently from the underlying geometry used for design. Finite element models have largely been created using geometry which was manually entered, and to a large part, redundant with the geometry used for design purposes. Lately, advances in solids modeling and FEM software has enabled engineers to achieve significant productivity gains and unparalleled ease of use by bridging the gap between these two disciplines.

Introduction/Historical Perspective

For over 30 years the finite element method has been recognized as the de-facto standard for the analysis of arbitrary geometries for a broad class of problems. In the beginning the lack of computer horsepower precluded the analysis of models with many degrees-of-freedom. The limited size of models (a few hundred nodes and elements) did not present a big obstacle, with most users "content" to enter nodal coordinates and element connectivities by hand.

As time progressed and the sophistication and speed of the hardware and analysis software increased, the manual creation of finite element models quickly became a tedious chore. Invariably, the analysis could not be performed in a timely enough manner so as to affect the design. Although we had progressed from manual "meshing" to mapped meshing techniques the creation of the geometry required by the mapped meshers still represented the bottleneck in the design and analysis process. Analysts would typically take a dimensioned engineering drawing and hand input the required geometric entities to their mesher, making many simplifications to help resolve time-to-analyze issues. Although this was a tolerable situation for 2D problems, constructing 3D models usually required many months of work and was, therefore, seldom done. In reality, the productivity problem was further exacerbated by the geometric simplifications which had to be done. Often, when a detailed analysis was required for a small feature, a "staged" analysis was performed. The analysts would create an overall model which initially neglected the area of interest and would then create a *second*, smaller model, of the required feature using the results of the overall model as boundary conditions. If the analysts were lucky enough to perform the analysis in time to affect the design, and a redesign was put into place driven by the analysis results, they would then be faced with the daunting task of performing the whole process all over again.

With the advent of CAE/CAD/CAM systems some of the tedium of recreating the geometry went away. Although, the geometry created by these systems was seldom good enough to do meshing, it at least started to form the foundation to make life easier. The real problem with first generation CAE systems was in the type of geometry that was used. More often than not only wireframe entities were available, and if you were lucky, surfaces. For most general 3D purposes the geometry was very ambiguous and only slightly relieved the meshing burden and did nothing for the remainder of the finite element process, i.e., the addition of loads, restraints, material properties, etc. With these systems we were still forced to deal with the artifacts of finite elements, i.e., nodes and elements, instead of dealing with the more natural engineering entities such as vertices, edges, faces and volumes. Furthermore, we were restricted by the particular nuances of the analysis program that was employed. For example, if the analysis code would only accept a constant value of pressure over the entire element face we were forced to align meshes to load boundaries and make approximations to the actual loading. An ambitious user could compute equivalent nodal forces by hand, but this required a fairly good knowledge of general finite element principles, such as shape functions and numerical integration.

The way to resolve most of these issues is through the use of solid modeling to create the geometry and couple this process with a highly integrated finite element modeling and analysis system. Solid modeling has a strong appeal for design activities because it captures a mathematically complete description of the design geometry. When incorporated with modern computer graphics systems which are capable of displaying shaded, hidden surface removed images, solid models allow the design engineer to easily visualize his design concepts. When an integrated FEM/FEA capability is added, the user now has available a powerful, easy to use capability which overcomes many of the previously stated problems. Since the finite element entities are treated as standard attribute data attached to the geometry, a user can work on what makes sense from an engineering viewpoint and have the computer deal with the nuances of the finite element process.

What Can be Done Now

Solid Modeling

The current solid modeler used in the MSC/XLplus system is one based upon the ACIS kernel developed by Spatial Technology. It is a precise (as opposed to facetted) modeler which simultaneously supports wireframe, surface and solid geometries. Wireframe geometry is typically used to describe profiles which are swept or revolved to generate solids or surfaces. Solid primitives can be combined, that is, unioned, differenced, and subtracted, to form increasingly complex primitives or parts. The user interface follows a hierarchical paradigm which consists of dialog boxes, pop-up and pull-down menus that make the system easy for the casual user to both learn and operate. An underlying command language supports macros for automating frequently occurring sequences of commands for parametric design studies. The interface is feature oriented to allow a more natural engineering approach rather than a mathematical one. For example, a hole can be drilled through an object, or to a specified depth, with a user specified bottom. A feature can be quickly repositioned, altered (stretched, resized), or deleted. Features and primitives are positioned in space through the use of a wide variety of built-in or user defined coordinate systems.

For the purposes of performing design studies the solid modeler is also fully parametric. Users can define those pieces (features, primitives, bounding curves, etc.) which they desire to modify by assigning a unique variable to the relevant geometric parameter. When one of the variables is modified, such as the radius or location of a bolt hole, the system automatically updates the entire model and displays the result to the user for verification. A Design Rule Processor (DRP) is an integral part of the system and enables users to solve simultaneous non-linear equations which link the geometric parameters to "environmental" factors, material characteristics, cost and other geometric information.

The geometric modeler has been constructed such that the user has complete freedom as to the amount of detail which is to be included. For the purposes of pure design, facilities exist which allow complex fillets, chamfers, rounds, etc. to be specified. If the solid model is being generated solely for analysis purposes, and small features can be ignored, a user simply never adds them, or uses the parametric capabilities to remove existing ones.

FIGURE 1 shows an example of a simple solid model which can be created with MSC/XLplus.

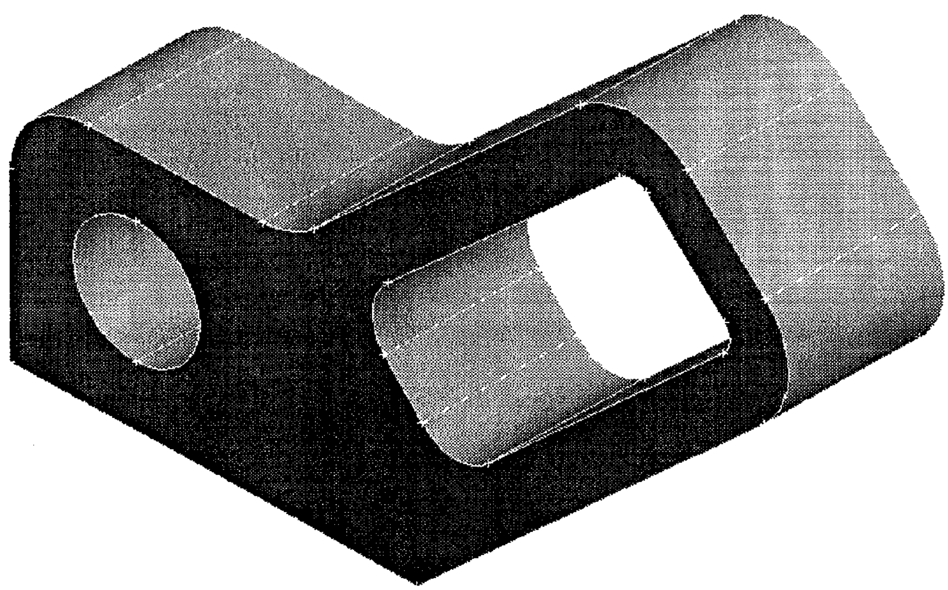

Figure 1

Free Body Diagram (ENVIRONMENT Module)

During the design process, the solid model is the "real thing." It carries a geometrically complete description of the object which will eventually be manufactured. Finite element models, on the other hand, are abstractions whose only purpose is to simulate the behavior of the design.

To achieve a high degree of productivity and ease-of-use the finite element modeling systems must be able to understand the data structures used for the solid model and possess "natural" user interfaces. While most design engineers do not understand the subtleties of finite element modeling, they are very comfortable with the use of free body diagrams to express how their designs are fixtured and loaded.

Traditional free body diagrams show the external loads acting on the body along with reactive forces acting at the boundaries of the body. In most cases, design engineers don't know what the reaction forces are. Instead they know how the body is restrained (fixtured). The MSC/

XLplus system extends the concept of a free body diagram to include restraint and boundary condition information.

Once the solid model has been created the user applies load and restraint data directly to the solid model. The MSC/XLplus software allows the user to attach load and restraint data directly to the vertices, edges and faces of the solid model without any reference to the finite element mesh which will eventually be generated. The user is provided with a rich set of tools that allow constant, linear, quadratic and piece-wise linear variations of the loads and restraints to be specified and then applied to either the entire, or a portion of a geometric edge or face. Concentrated point loads can be specified anywhere within the model, without reference to any underlying finite element node location. The engineer has complete control over the coordinate system to be used for both input and output and is provided with complete graphical feedback delineating direction and magnitude.

FIGURE 2 demonstrates the results of adding a quadratically varying pressure to the face of a solid model.

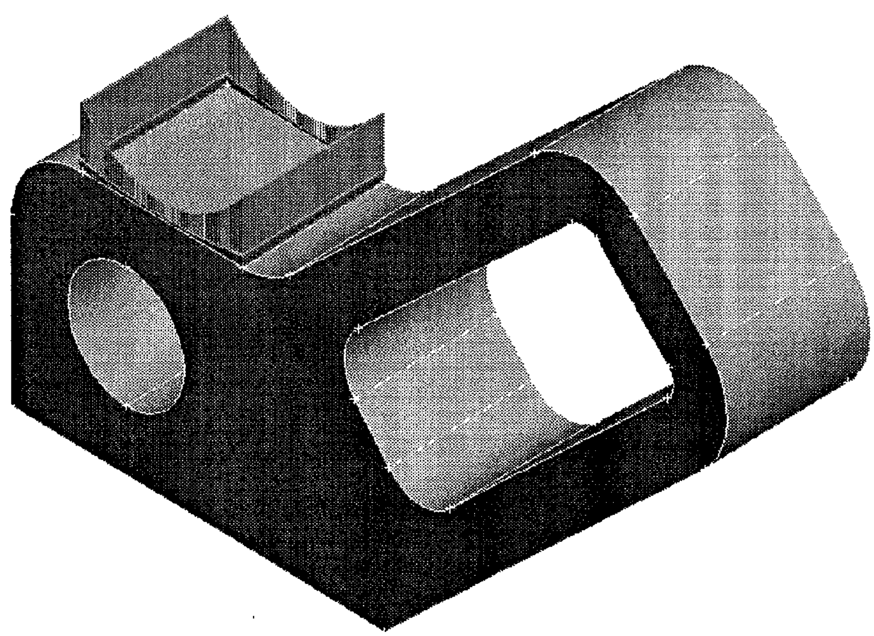

Figure 2

Finite Element Model Creation

Once the load and restraint information has been added to the solid model, the finite element modeling module is used to automatically generate the equivalent finite element model. All the data required to generate the FEM has been attached to the solid model. In addition to geometry and topology data, the MSC/XLplus solid modeler also carries material information. The material data attached to the solid includes all of its mechanical and thermal properties. Thus, information such as Young's Modulus and Poisson's Ratio are immediately known. If the material properties are ever changed for the solid they are automatically applied to the finite element model.

To make the creation of nodes and elements as easy as possible, MSC/XLplus has been implemented with an automatic mesh generator based on quadtree and octree technology. The geometry created by the solid modeler is used directly as input to modules which generate a mesh of either linear or quadratic triangles or quadrilaterals in 2D, or tetrahedrons in 3D in a fully automatic manner. To generate a mesh, the user simply picks the solid(s) to be meshed, gives a default element size and selects the "mesh" menu item. Optionally, the analysts can attach mesh constraints to force local refinement in critical areas. Here again, these mesh constraints are attached directly to vertices, edges, faces and regions of the solid. FIGURE 3 shows the result of a mesh automatically generated with MSC/XLplus.

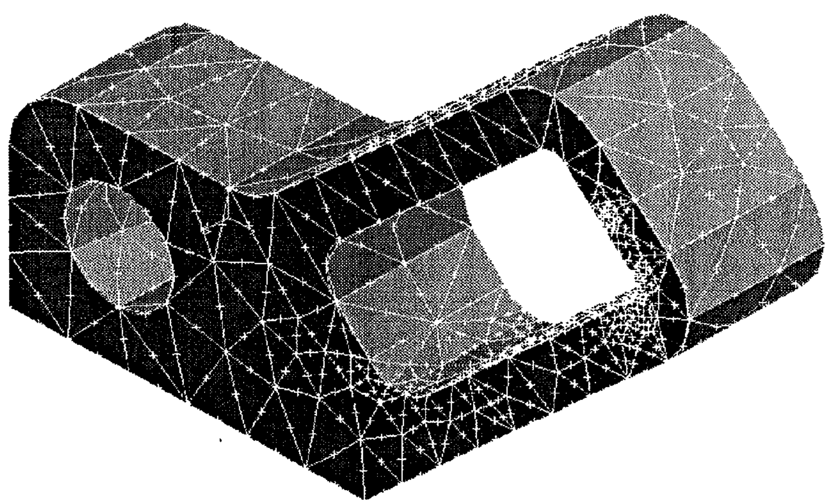

Figure 3

In addition to the fully automatic generator, MSC/XLplus also has the more traditional mapped mesher for the creation of pentahedral and hexahedral finite element models. As with the automatic method, the mapped mesher works directly off the geometry. Since a Coon's Patch (transfinite mapping) technique is employed by the mapped mesher it is capable of handling geometries with relatively complex shapes. To provide maximum flexibility the MSC/XLplus system allows both automatic and mapped meshes to exist in the same model. Capabilities exist which allow an automatic mesh of quadratic tetrahedrons to be butted up to a mapped mesh of linear hexahedrons with constraint equations automatically generated to tie the nodes of the tets to the nodes of the bricks. To round out the capabilities, fully manual addition, deletion, and modification of the mesh is also available.

After a mesh is created the loads and restraints that are attached to the geometry must be transformed from their geometric based representations to the corresponding finite element ones. To accomplish this, the engineer simply selects the desired load and restraint case(s) and instructs the system to perform the required mapping. The knowledge of how to convert information attached to the solid model to the equivalent finite element entity is embodied in the algorithms of the finite element modeling program. The system queries the geometric based data as to the variation that was specified, the coordinate system definition, the type of geometry it was applied to, the span of the geometry, and the specified finite element model type and automatically creates the appropriate FEA entities. These FEA entities can be obtained by simply sampling the corresponding geometric ones, or by performing work equivalent load calculations. FIGURE 4 shows an example of the transformation from a geometric based load to finite element ones.

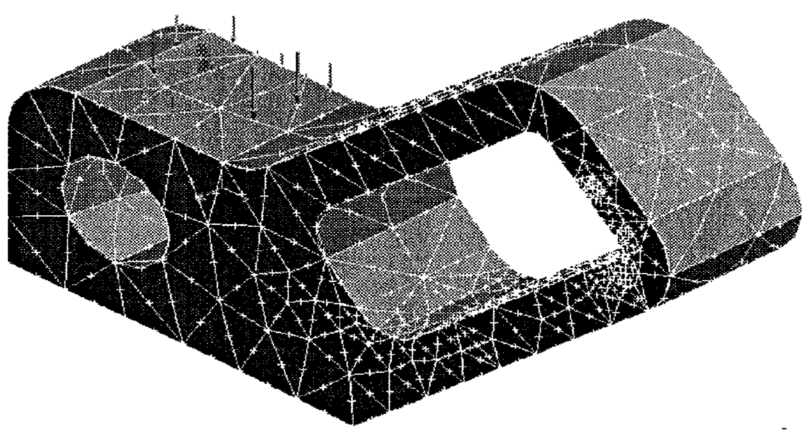

Figure 4

Once the loads and restraints are generated, the finite element model is complete and can be analyzed. The total time to convert the free body diagram to a finite element one is typically done within a few hours to a few days depending on the complexity of the underlying geometric model.

Postprocessing

Once an analysis has been performed the analyst has a complete suite of graphical postprocessing tools available. Included in this list are deformed geometry and contour plots, XY graphs and printed reports. Facilities exist to allow users to perform error estimations and manipulate the raw data to form new, derived information. A capability to animate deformed geometry along with a unique dynamic section plane showing continuous updates of the stresses on the plane are also available. A complete set of averaging options, used in conjunction with contour plots, is also available to the user, offering greatly enhanced graphical capabilities. The entire postprocessor was written as an "open" system to allow users to import data into the postprocessor, which may not be "understood" by the preprocessor. This facility permits the analysts to identify any data which can be described as a scalar, vector or tensor to be brought into the system and manipulated using the standard tools. All the engineer has to do is to give the basis of the data (node or element), what the associated units are, and whatever meaningful name or tag that the user wishes to identify the information with.

FIGURE 5 shows a typical contour plot generated by the MSC/XLplus system.

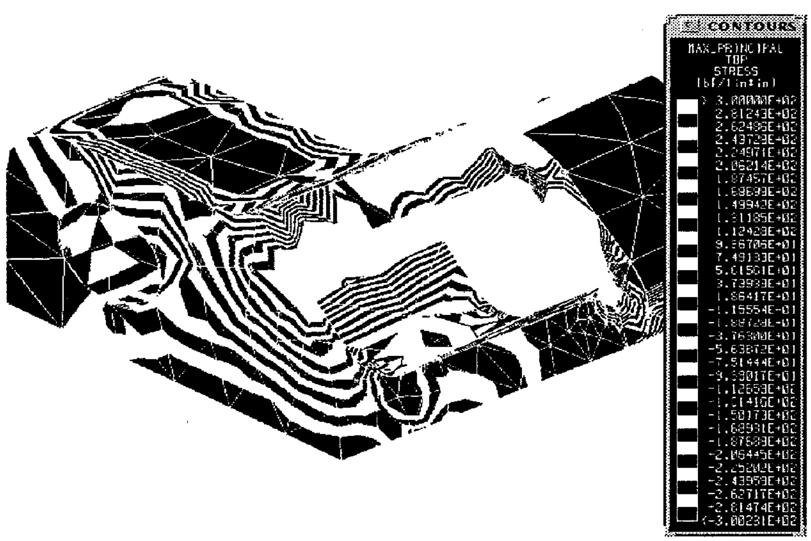

Figure 5

The Short Term Future

The following items represent work-in-progress that are expected to be released by the end of 1993.

"Meshless" Analysis

This project is aimed at removing the artifact of the finite element mesh from the user. It does not imply that a finite element mesh will not be used. The aim of the meshless analysis project is to take advantage of the geometric capabilities of the current MSC/XLplus system and put a wrapper around them so as to produce a seamless analysis function with which the user only works with the geometry. The salient features of this capability are:

- Addition of loads and constraints directly to the geometry
- Specification of an allowable error
- Automatic mesh, load and restraint generation
- Error estimation
- Automatic grid refinement based on the error
- Continuous loop which performs successive analyses until the error is within the specified tolerance
- Display of results showing only the underlying geometry rather than the finite element model
- Access to the finite element model if desired by the user

Shape Optimization

As the name implies this feature is meant to tie in with the shape optimization capabilities destined for version 68 of MSC/NASTRAN. Instead of dealing directly with finite element representations of design variables users can take advantage of the parametric features of the solid modeler with the system automatically converting these variables into MSC/NASTRAN terms. Anything which can be parameterized or defined with the DRP facility can be specified to be a design variable. Constraints, such as max stress, will be characterized by identifying the geometry rather than individual nodes and elements. Standard MSC/NASTRAN capabilities such as variable linking will also be supported, but with a graphical orientation and user interface. When new values of the design variables have been computed by MSC/NASTRAN they will be returned back to the MSC/XLplus system to update the geometric model.

This project will eventually incorporate the meshless analysis function to ensure that the optimization procedure is utilizing a finite element mesh which is accurate.

Miscellaneous Projects

The following are smaller scale projects which are destined to incorporate existing, as well as new capabilities of MSC/NASTRAN.

- Support for eigenvalue buckling

- Support for current, and version 68 heat transfer capabilities
- P-elements
- Material non-linearities
- Geometric non-linearities
- Substructuring

Further Out

The following projects are those that have been proposed as items which will put the full power of MSC/NASTRAN into design engineers' hands, while offering an easy to use interface which will allow them to take advantage of that power.

Hexahedral Mesh Generation

The project is currently underway and represents an effort to develop a fully automatic mesh generator which produces a hexahedral mesh rather than one consisting of tetrahedrons. The interface to this capability would be identical to what is currently done, but will produce the superior performing brick elements.

Analysis Model Simplification

Similar to the auto hex meshing project, elements of this project are also underway. The purpose of this effort is to recognize those 3D solid models which can, and should be treated as shells or beams and then automatically extracting out the necessary physical properties (thicknesses, cross sectional areas, moments of inertia, etc.) from the underlying solid representation.

Automatic Tying of Distinct Parts

When dealing with assemblies, or a collection of disjointed parts there must be a method by which load is transferred between the individual pieces. Building upon the current ENVIRONMENT capabilities, users will be able to identify those geometric areas of a model which they desire to have connected. These connections can take the form of gap elements, contact surfaces or constraint equations. The idea is to allow the user to point to the geometry, identify the type of tie and then have the system automatically generate the appropriate information.

Full Substructure Support

This capability is also built on the idea of assemblies and will allow the user to identify individual substructures by simply choosing a geometric entity. Similar to the representations used by geometric modeling systems to graphically show a hierarchical assembly tree, this capability will provide a "superelement tree" and will allow an analysts to pick and choose those substructures which are to be used for an analysis with the system automatically providing the correct identification of external grids.

Conclusions

The integration of two powerful concepts, solid modeling and FEA, has resulted in a new system which offers distinct advantages over traditional design and analysis software packages. The ability for design engineers to work in terms which are natural to them has provided them the ability to use powerful analysis tools in a very easy and user friendly fashion. Large increases in productivity, translating into decreased time to market, have been gained. There exists the future promise of having large scale systems which can produce required engineering analyses in very short time frames which truly allow for "design by analysis."