Redefining the Process of Airframe Finite Element Model Development Using MSC/SuperModel

Michael Farley
Section Head of Methods & FEM
Raytheon E-Systems Waco
7500 Maehr Road, MS 1134, Waco, Texas 76705

Editor
Kirsten E. Husak

ABSTRACT

To develop, in less than one year, a MSC/NASTRAN 747SP finite element model from scratch to support the SOFIA (Stratospheric Observatory for Infrared Astronomy) program is an effort requiring significant planning.

Model development is facilitated by dividing the airframe structure into submodels enabling parallel development of each submodel within separate MSC/PATRAN databases. Directing this effort requires configuration control and file management to maintain the overall pedigree of the model, which includes section properties, history files, drawings used and assumptions made. This paper discusses how MSC/SuperModel is being used to redefine airframe finite element modeling at Raytheon E-Systems Waco.

INTRODUCTION

Traditionally airframe finite element models were characterized by bulk data files which provided the master definition of the airframe structure. The bulk data file was annotated with comment cards to help maintain the pedigree, assumptions, and history of change to the master finite element definition of the airframe structure. In the past, many airframe finite element models were developed without the aid of sophisticated pre-processors. These models existed as ASCII text bulk data files which were easily ported between platforms and could be archived, or recovered from an archive, with small risk of data corruption. With the advent of pre- and post-processing applications, the bulk data file was read into such an application for viewing and incorporating changes, but the master definition of the airframe finite element model was still the province of the bulk data file.

Aerospace companies have used commercially available applications for pre- and post-processing but have found them noticeably lacking in areas relating to airframe finite element analysis, as early commercial software was not developed with airframe structure in mind. Commercial software developers placed emphasis on the pre-processing aspect of finite element development (for instance, if topological and geometrical congruence was maintained on the geometry level, theoretically the entire finite element mesh could be generated with one mesh command). The post-processing side of these commercial applications did not adequately address the need for recovery of internal loads from the finite element model. Some of these applications were impressive in scalar post-processing, but a contour stress plot of fuselage skins is not very useful in a detailed stress analysis.

Some aerospace companies went so far as to develop in-house finite element pre- and post-processors to support analysis of airframe structures. These applications excelled at viewing and recovering internal loads to be used for subsequent hand stress analysis, but companies engaged in such development were now involved in application design. This involvement required significant human resources dedicated to the development, documentation, maintenance, and support of finite element application software. Many companies have found dedicating human resources in this manner to be cost-prohibitive and have sought the partnership or have developed relationships with CAE companies in order to reduce overall operating costs and enhance productivity.

In the development of the finite element model (747SP) which supports the SOFIA program, pre-and post-processing and data management are being handled with great success by MSC/SuperModel. This paper illustrates the use of MSC/SuperModel in development of the SOFIA model.

FILE MANAGEMENT

The MSC/SuperModel application includes a system (FileManager) which can be used to manage MSC/PATRAN databases and all files relevant to model development and analysis, including geometry files created with CAD tools. These files are maintained in a site-defined file hierarchy. The hierarchy was developed to provide structure to the finite element model development process: by maintaining a comprehensive list of all versions of all models for all programs at a site, the hierarchy provides configuration control and establishes the pedigree of the master finite element model (see figure 1).

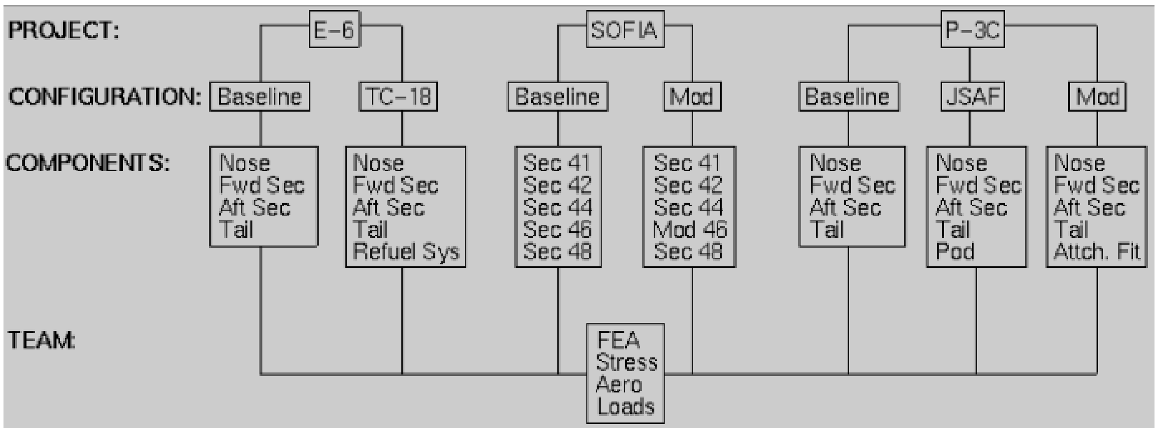

Figure 1: MSC/SuperModel File Hierarchy

CONFIGURATION CONTROL

Several airframe finite element models have been developed or are in the process of development at Raytheon E-Systems Waco. Each of these models has several different versions, some of which are years old and have changed hands a number of times. A lack of readily available documentation indicating changes to each version of a model can lead to a lack of faith in the validity of the internal loads developed by the model for current stress analysis.

For the SOFIA 747SP finite element model, it is a requirement that the history of the model assembly be properly maintained, that all documentation used to develop the model (including all engineering assumptions made) be electronically associated with the model, and that CAD part files which define the model geometry be electronically associated in the same fashion. This data management requirement means that an integrated method of data maintenance is needed, and MSC/SuperModel is well suited to perform this function.

The MSC/SuperModel FileManager ensures that all data (including CAD geometry) is resident in the repository of the MSC/SuperModel file hierarchy. The file repository is an allotted space on a hard disk which houses all files under the control of the FileManager.

When working on a model in MSC/SuperModel, the user is prompted on exit for input to the history file which is maintained within the MSC/SuperModel hierarchy and can be referred to at need. A typical example of how a version change is maintained in MSC/SuperModel is shown in figure 2.

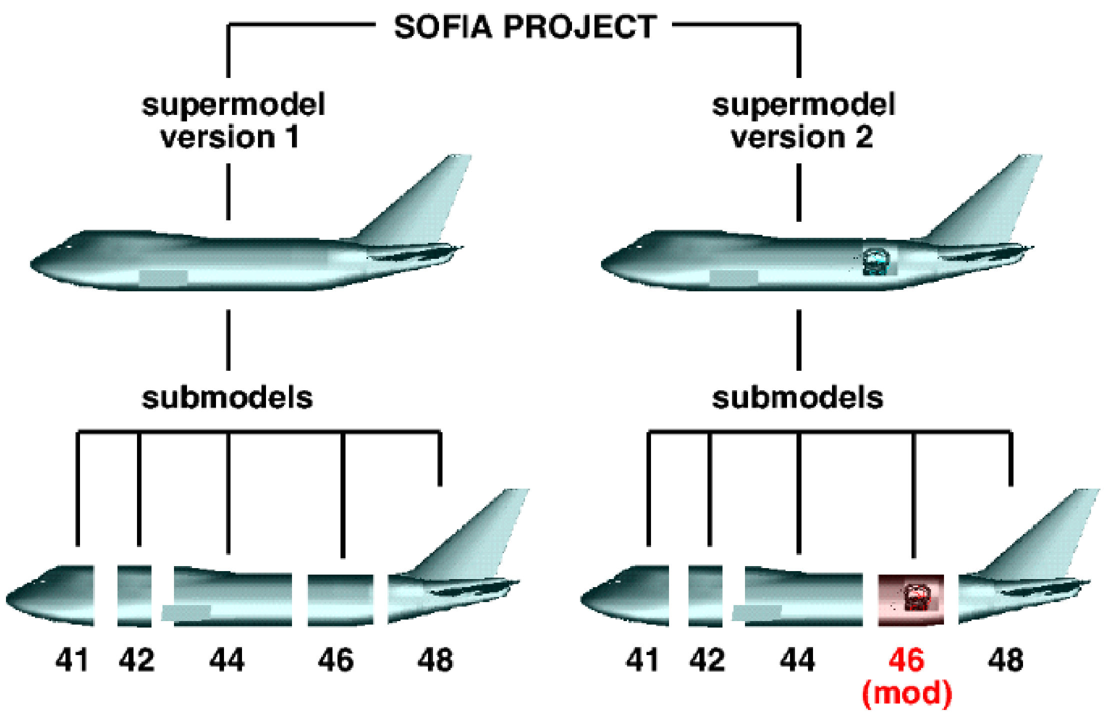

Figure 2: A Version Change Maintained in the MSC/SuperModel Hierarchy

SUBMODELS

Development of an airframe finite element model requires division of engineering labor. Models are often built by two or more companies remotely located, which model specific sections of the airframe structure. These sections may also be sub-divided and the resultant sub-sections assigned to different personnel. Such division of a model requires that interface points, unique elements, grid ids, and property ids be defined and understood by all parties. A lack of communication or poor logistical execution can be identified by interface points which do not match, conflicting ids for grids, elements, or properties, and inconsistent property values.

For the SOFIA program the Methods and Finite Element Group is using MSC/SuperModel to address the above issues. Each section of the 747SP finite element model is developed in a distinct MSC/PATRAN database within the MSC/SuperModel hierarchy, and these separate databases are integrated into one SuperModel by MSC/SuperModel. Figure 3 illustrates the integrated use of distinct MSC/PATRAN databases (or submodels in MSC/SuperModel terms) within MSC/SuperModel. This integration allows for division of effort, and the problem of an interface and consistent property values between separate submodels is addressed by assigning templates in the individual MSC/PATRAN databases. These templates are 'hard-wired' into the database and ensure that all modelers are working with the same interface and properties. The templates in use for the SOFIA program are shown in figure 4.

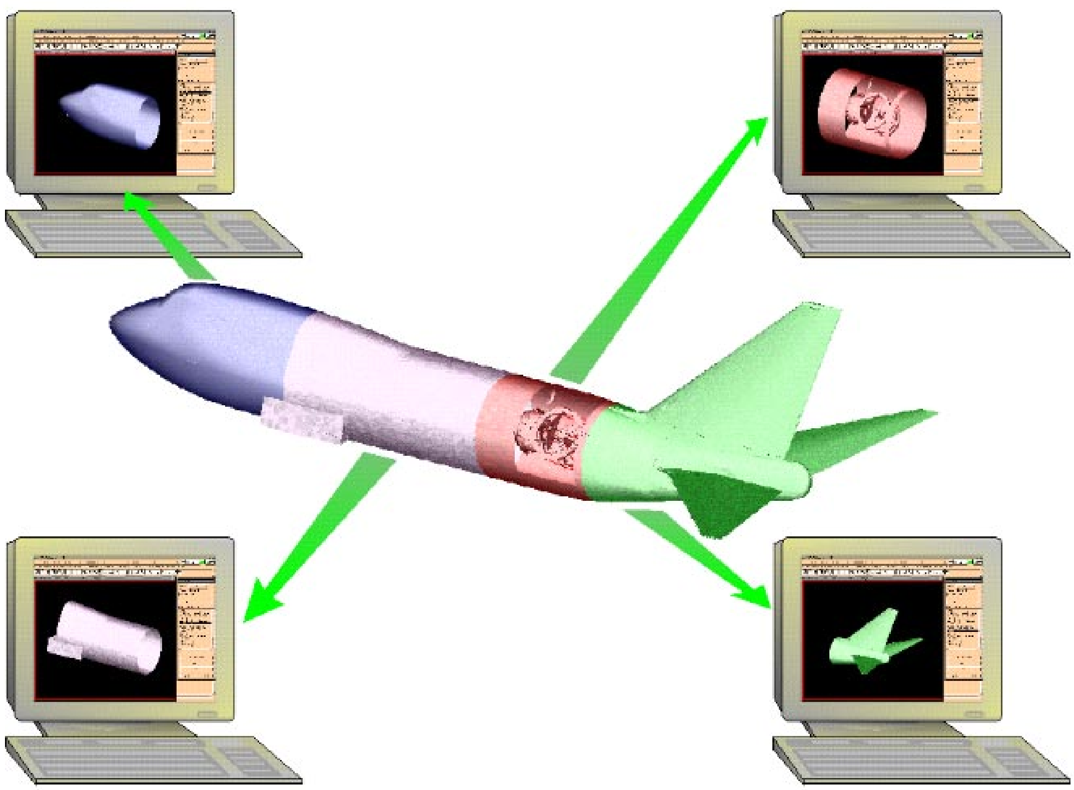

Figure 3: Distinct MSC/PATRAN Databases as Submodels in MSC/SuperModel

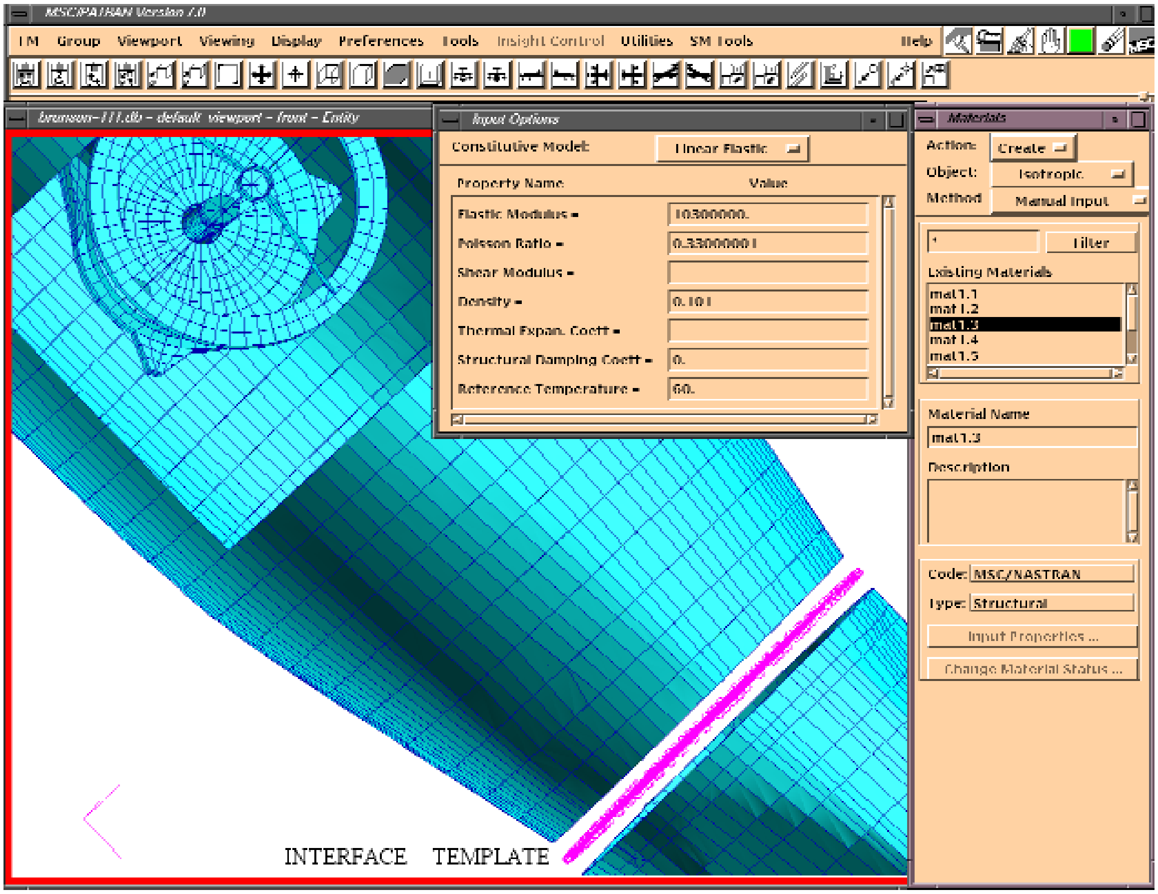

Figure 4: SOFIA Material Property and Interface Template

MODELING PHILOSOPHY

Airframe finite element model numbering schemes were devised to help locate elements and grids by their ids: if a grid has an id prefixed by 477, the grid is located at frame 477. In cases where a numbering scheme is a requirement, the merging function of MSC/PATRAN with MSC/SuperModel can maintain the scheme within the MSC/SuperModel hierarchy, but MSC/SuperModel places emphasis on the graphical user interface (GUI) as a performance environment for all aspects of the finite element analysis process, reducing the importance of element ids, grid ids, et cetera. This concept has merit because when a stress analysis of an airframe structure is performed, the analysis is in terms of specific components of the structure (with which no id is associated). For instance, when considering an airframe structure, visualization is in terms of frame station 477, stringer 27 left between two frame stations, or fuselage section 46.

The MSC/SuperModel master database graphically represents all features of the airframe, which means that the model can be queried on any parameter of interest (including property values and internal load results) which defines the airframe structure or results.

This graphical representation renders element ids less important. The finite element model can be thought of in terms of its real structure when referring to the database model, both for MSC/NASTRAN job submittal and for results post-processing (including max/min results, shear and bending moment checks, freebody forces, and configuration control). This point bears repetition: due to its graphical characterization, MSC/SuperModel allows the finite element model to be thought of in terms of its actual structure, rather than as a series of element ids.

GEOMETRY DEVELOPMENT & CAD ACCESS

The CAD system (Pro/ENGINEER) was used to define much of the geometric shape of the finite element characterization of the airframe structure. The definition of the frames, keel beam, torque box, wing section, and floor structure were all constructed in the CAD environment. The relationship between MSC/PATRAN and Pro/ENGINEER enables MSC/PATRAN directly to access the Pro/ENGINEER database and to import the geometry in its native CAD form (see figures 5 and 6). The technique of employing a CAD system to develop geometry for finite element models creation has technical merit for two reasons: 1) CAD systems are designed to develop geometry, and therefore in general are more efficient tools for geometry development, and 2) development of CAD part files often precedes development of a finite model. In a concurrent engineering environment CAD geometry is defined as the original source of the Single Geometry Model (SGM). To state it simply: design, analysis and manufacturing use the same CAD geometry part file.

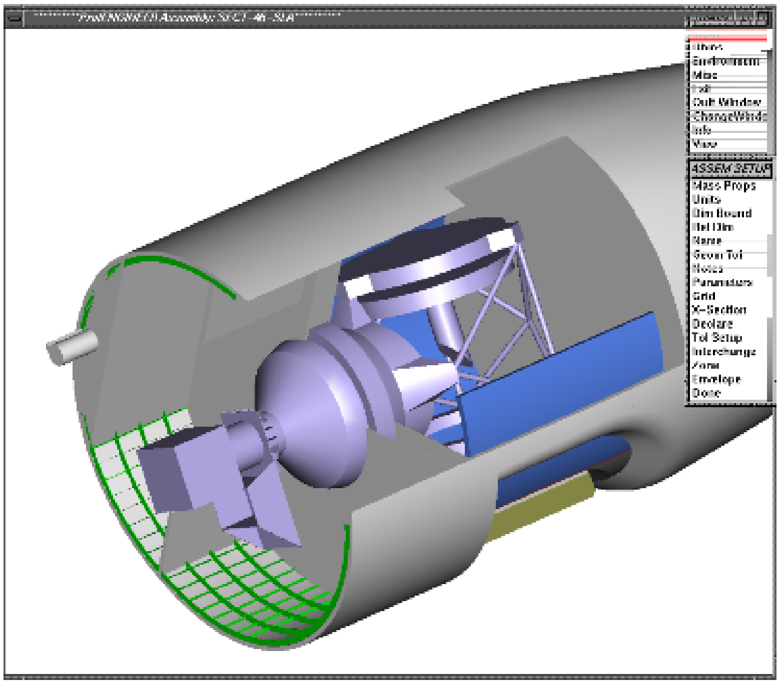

Figure 5: Pro/Engineer Assembly

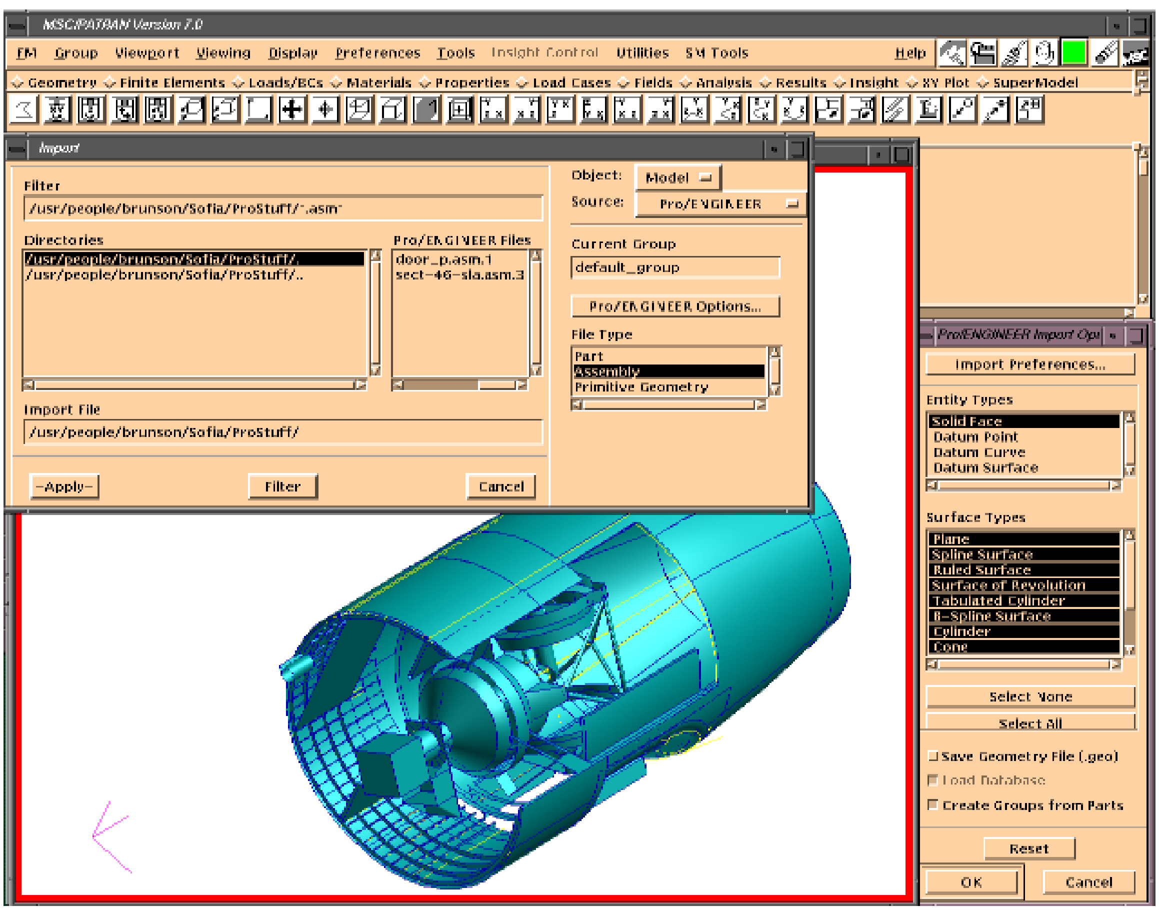

Figure 6: Pro/Engineer Assembly Imported into MSC/PATRAN in Native CAD Form

PROPERTY DEFINITION

MSC/PATRAN has a standard beam library, containing standard sections (hat sections, angle sections, etc.). When the user enters the dimension of the section, MSC/PATRAN calculates the section properties and creates the property card. Once the section properties have been calculated, they can be assigned to geometry, which enables modification of the mesh without redefinition of properties for each mesh change. Creating section properties using the standard beam library offers another advantage: on a query of the element for properties, MSC/PATRAN graphically displays the geometric shape of the section superimposed on the element. Non-standard parts can be added to the beam library; MSC/PATRAN will calculate properties for these sections based on a user-input algorithm, but will not graphically display them. Figure 7 shows the beam library form and illustrates a graphical display of a hat section associated with an element.

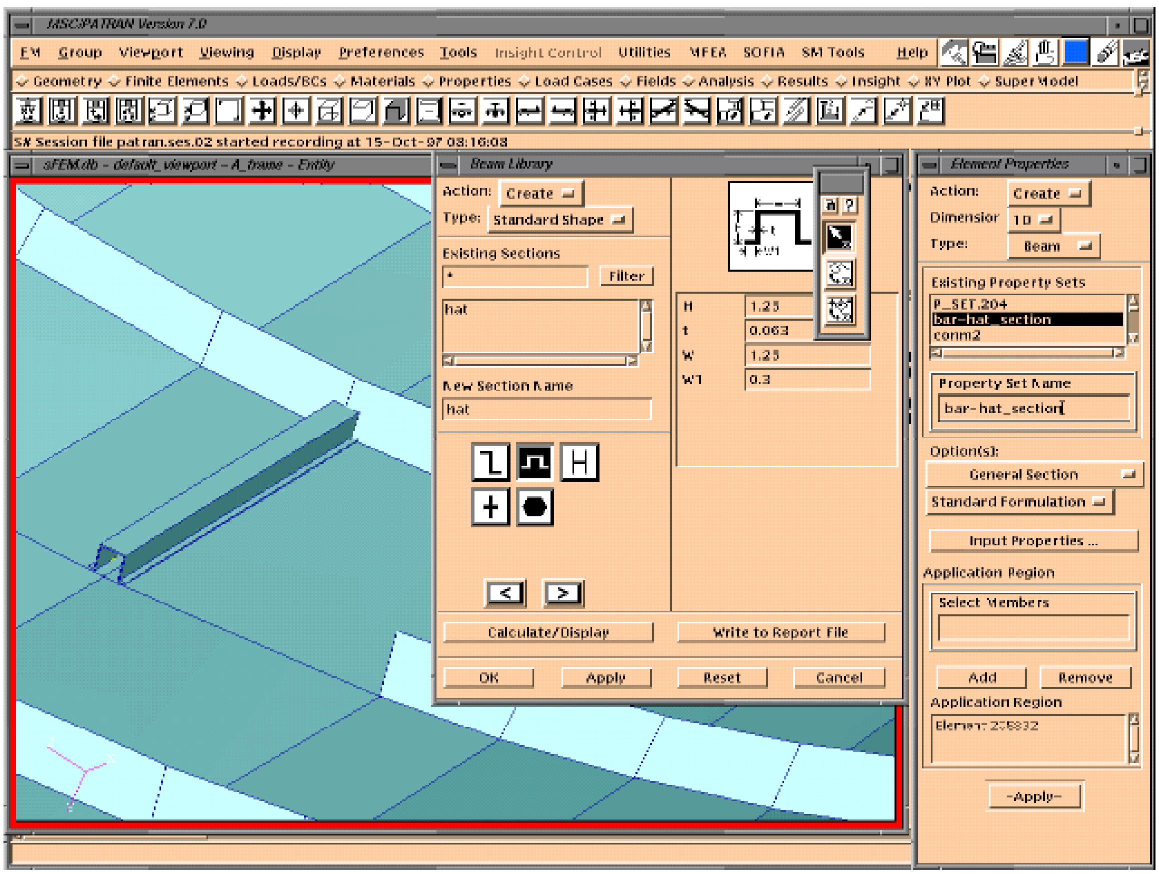

Figure 7: Graphical Display of a Hat Section Superimposed on an Element

EXTERNAL LOADS VALIDATION

Shear and Bending

MSC/SuperModel provides means to validate the external loads applied to the airframe finite element model database: the external loads are defined as load cases within the database, and shear and bending moments of load cases can be plotted and evaluated against original load case data. Load case validation is required to ensure that loads applied to the model are consistent with loads applied to the actual airframe. Figure 8 illustrates a shear and bending diagram produced with MSC/SuperModel.

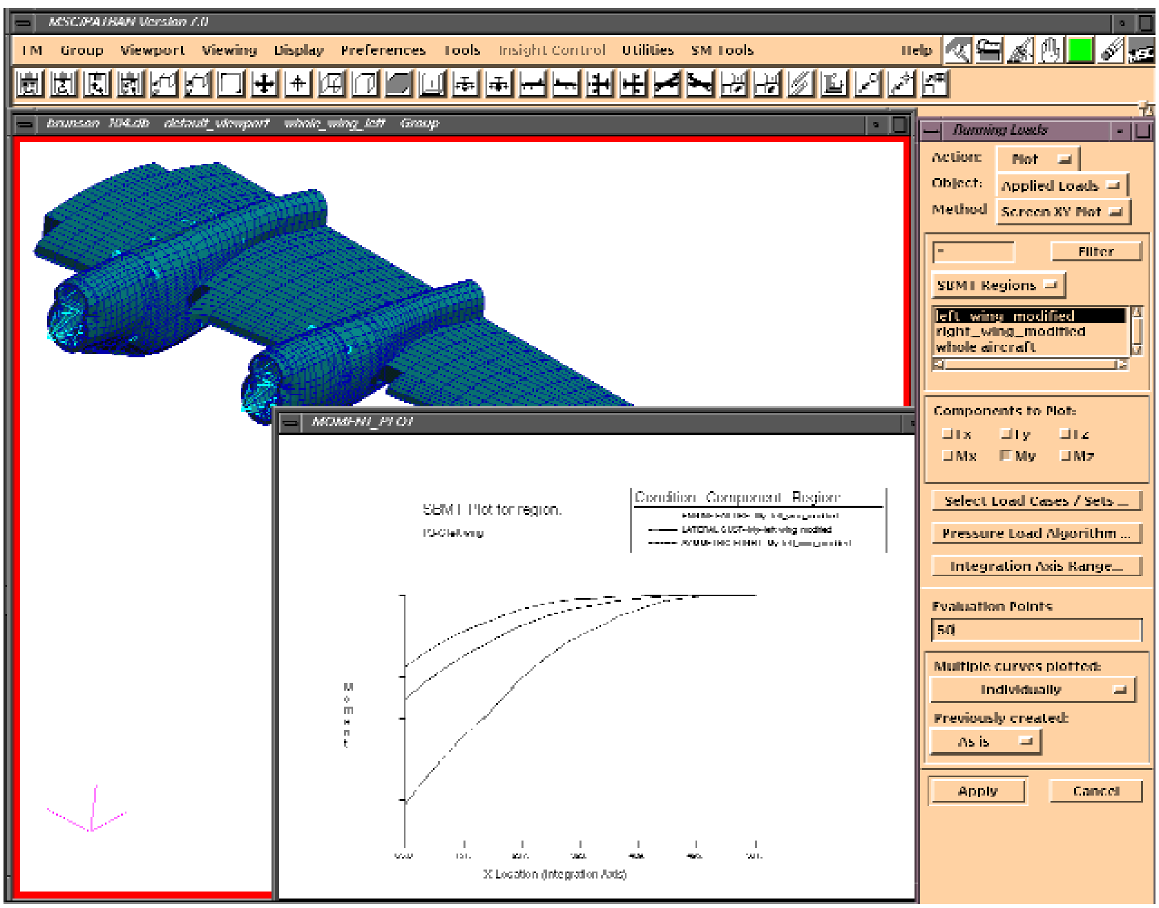

Figure 8: MSC/SuperModel Shear and Bending Diagram

Load Summation

The loads summation function in MSC/SuperModel considers all components defining a load case and displays load summation value. Such an evaluation is required for assurance that load cases are balanced (see figure 9).

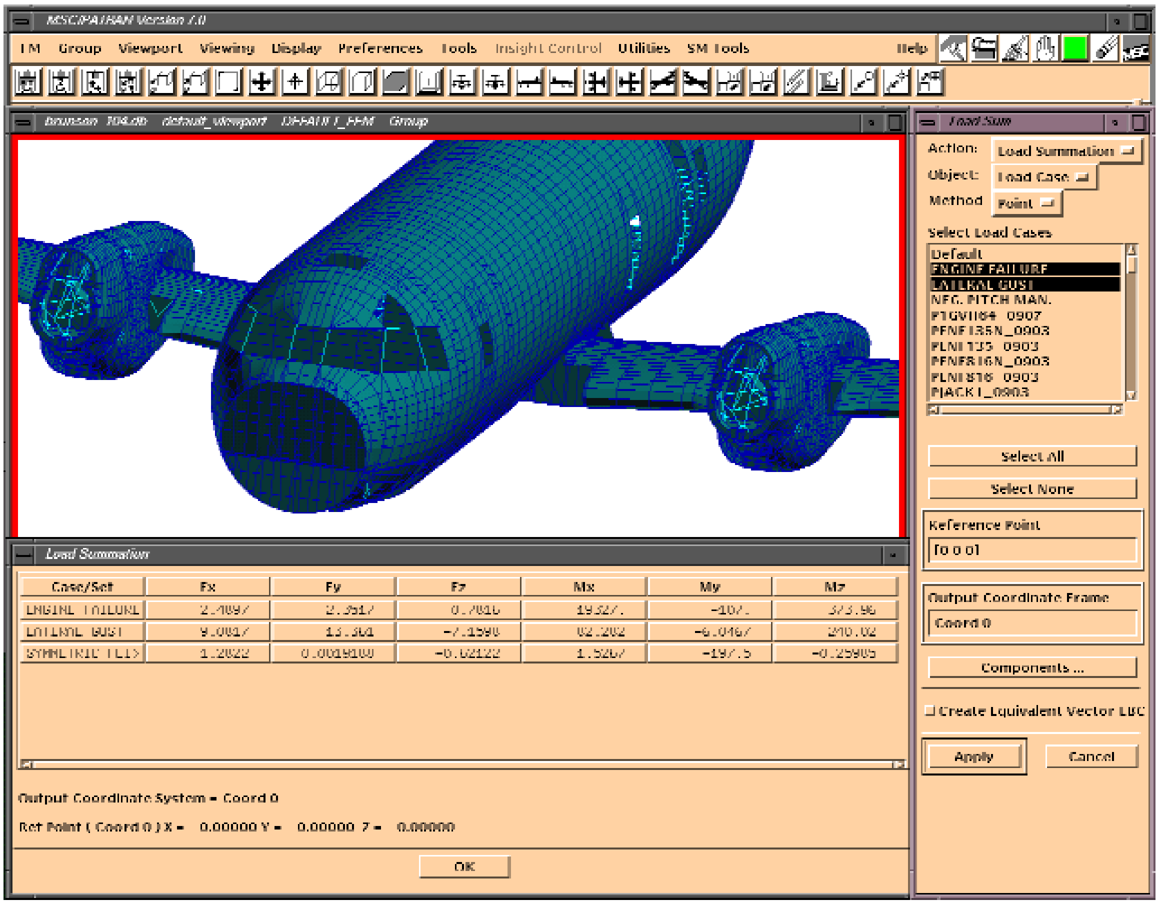

Figure 9: MSC/SuperModel Load Summation Form

JOB EXECUTION

A master definition of the airframe finite element model in database form enhances both job submittal and the results recovery process. The load cases, boundary conditions, and subcases are defined within this MSC/SuperModel master database, while exterior files (such as forces for load conditions) can be included in the MSC/NASTRAN job execution of the master database using the Include File form.

The model is submitted for execution to MSC/NASTRAN within the MSC/SuperModel application. MSC/SuperModel then assembles the separate MSC/PATRAN databases, performs an equivalence if necessary, resolves conflicting ID issues (if any), submits the job for MSC/NASTRAN execution, and returns results to the MSC/SuperModel database.

The job is submitted for MSC/NASTRAN execution by MSC/PATRAN Analysis Manager, which is integrated with MSC/SuperModel. Analysis Manager, a graphical user interface (GUI), queues and monitors MSC/NASTRAN jobs throughout execution. Analysis Manager provides functions which can set the memory usage level, check job status, and monitor CPU and disk utilization (see figure 10).

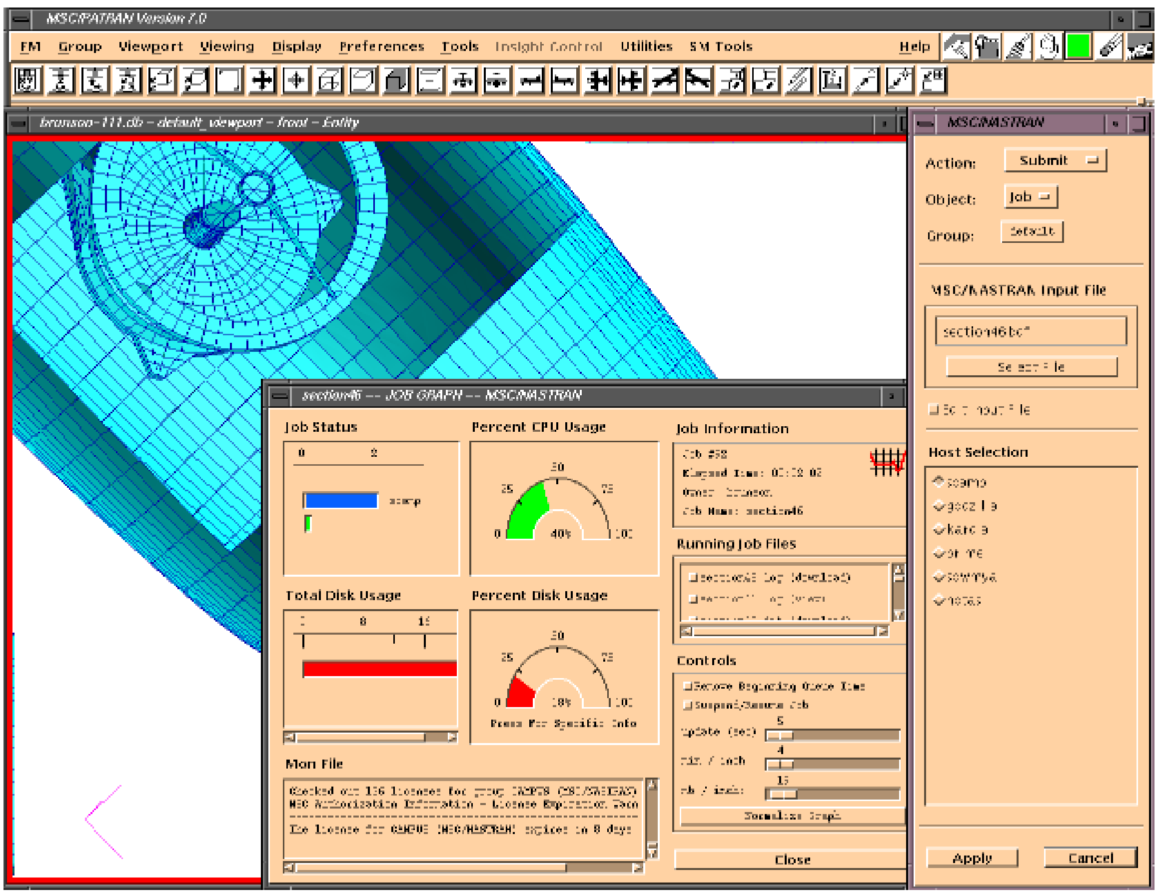

Figure 10: MSC/SuperModel Analysis Manager Form

POST-PROCESSING

After job submittal, results are automatically returned to the database and are accessible within the MSC/SuperModel hierarchy. This allows stress analysts to recover and display graphically any result from a master database, whose configuration is controlled by MSC/SuperModel. MSC/SuperModel includes conventional post-processing options, but also provides functions which are extremely useful in airframe analysis, such as max/min results sorting, shear panel plots, bar end loads, and freebody plots included in MSC/PATRAN (see figures 11-13).

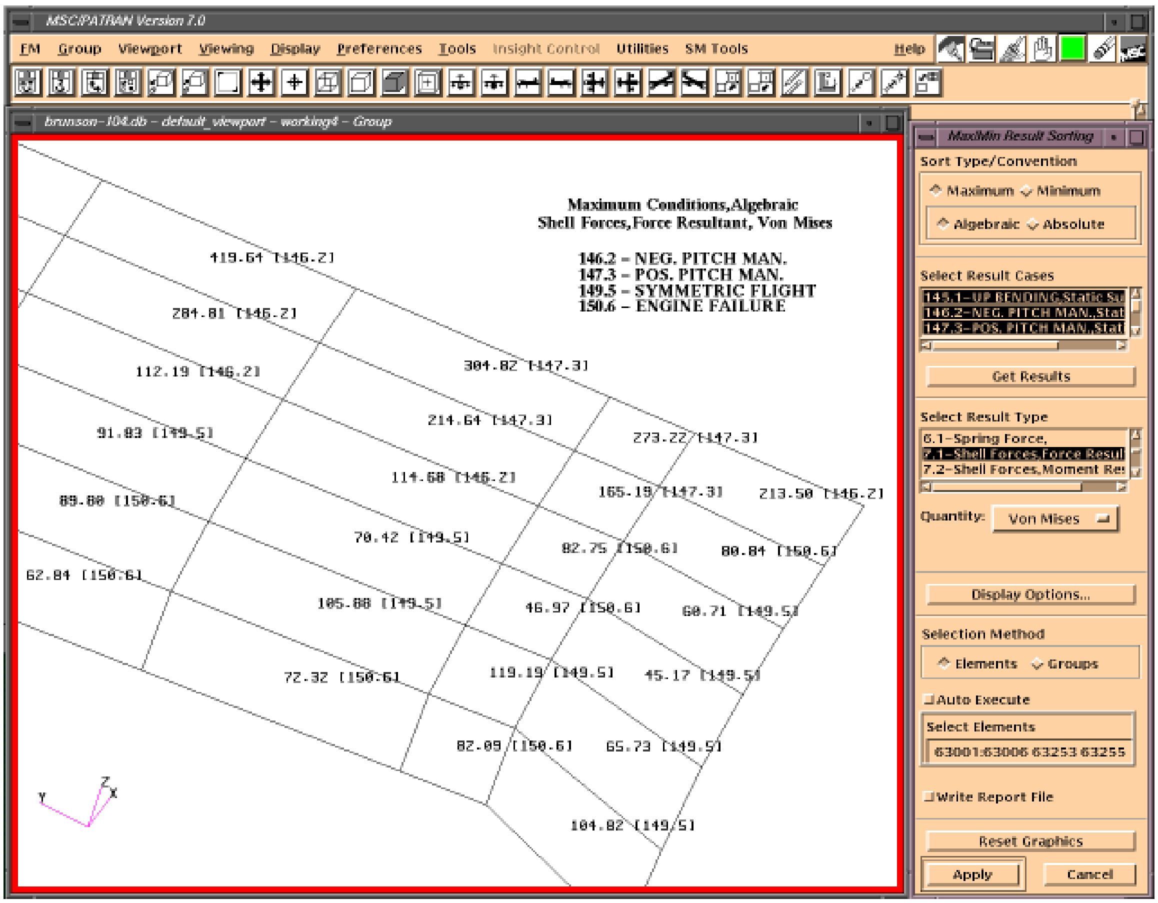

Figure 11: MSC/SuperModel Max/Min Sorting

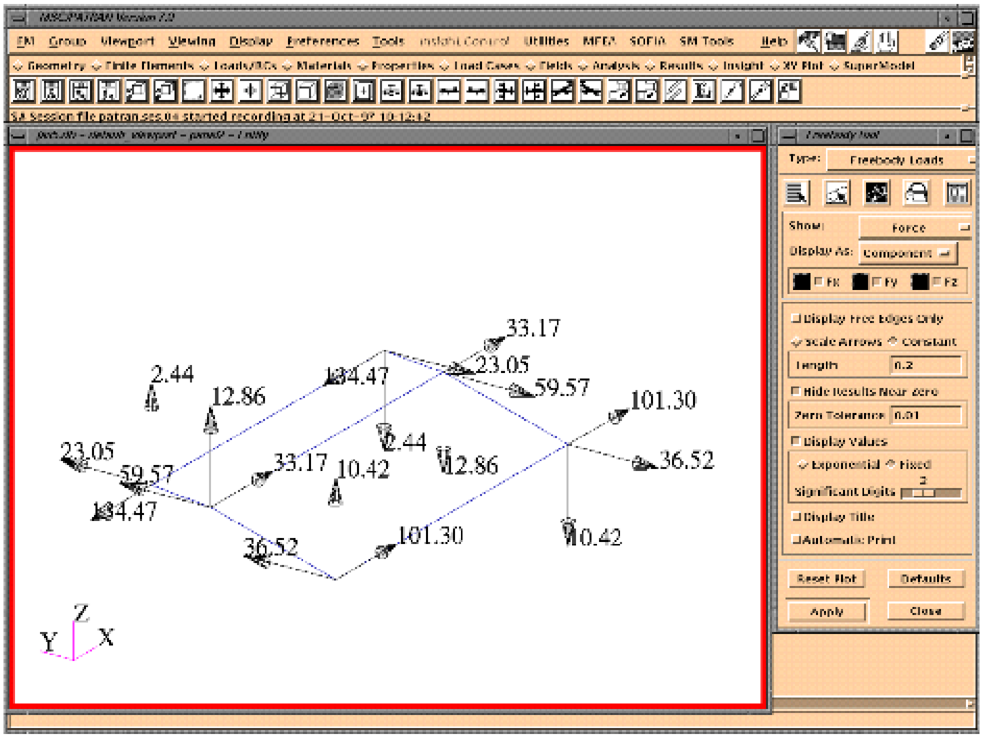

Figure 12: Freebody Plot

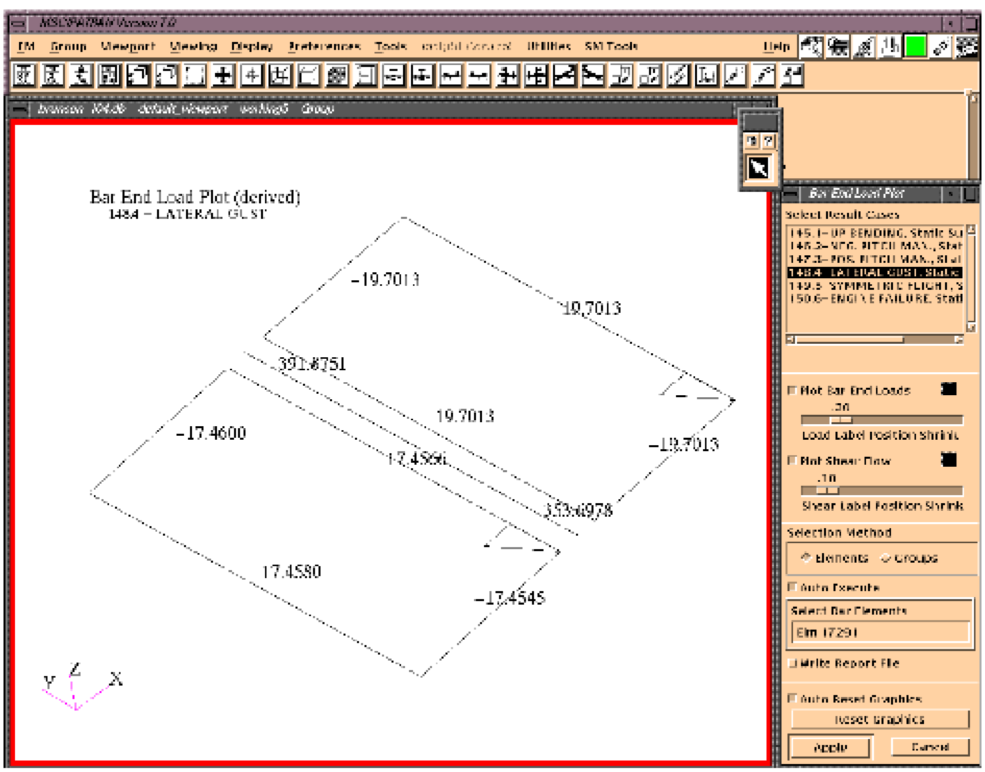

Figure 13: MSC/SuperModel Bar End Loads

SUMMARY

Developing a 747SP finite element model from scratch to support the SOFIA program afforded the Methods and Finite Element Group an opportunity to redefine the entire finite element process. The master definition of the SOFIA airframe finite element model is a MSC/PATRAN database within the MSC/SuperModel hierarchy. All functions relating to the finite element process were performed within MSC/SuperModel, including model development, external loads validation (shear and bending moments and load summation checks), and job submittal. Internal loads such as max/min sorting, freebody loads and bar end loads were recovered within the MSC/PATRAN database. MSC/SuperModel places emphasis on the graphical user interface in performance of all tasks involved in airframe finite element development and analysis, which enables the analyst to think of the model in terms of the true airframe structure. Critical to the overall process is configuration control of the finite element models, which includes maintaining the pedigree of the finite element database by electronic association of all data used in model development. Through its successful use in addressing these issues, MSC/SuperModel is helping to redefine the process of airframe finite element development and analysis.

ACKNOWLEDGEMENTS

I would like to offer special thanks to Lee Brunson and Mark Dowell, members of the Methods and Finite Element Group here at Raytheon E-Systems Waco, who developed all the images for this paper, and created the images and animations for the presentation.