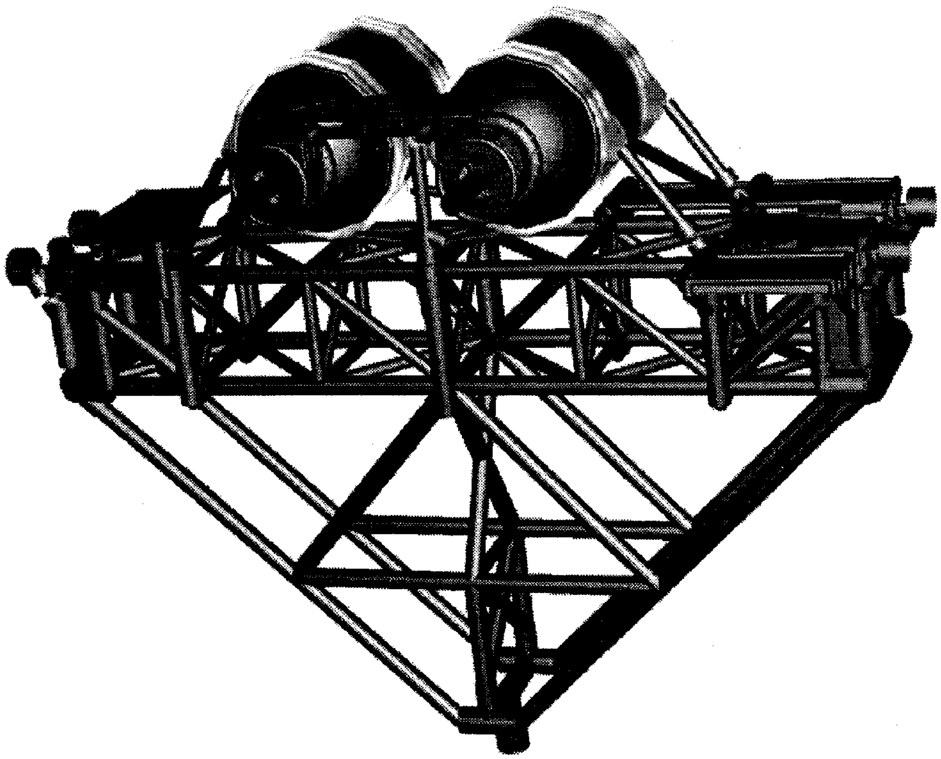
An Advanced Post Processing Methodology for Viewing MSC/NASTRAN Generated Analyses Results

Michael McMeekin
and
Paul Kirchman
Swales and Associates, Beltsville, Maryland 20705

Abstract

This paper presents an advanced post processing methodology for viewing analysis results generated from MSC NASTRAN normal modes analyses, and transient response analyses. In both cases the analysis results are added as a function of time to the basic finite element model geometries through specified coordinate transformations, creating a new deformed geometry at each time step. The final deformed geometries are rendered for each time step and animated through a public domain software utility on a personal computer for a final visualization of the analysis results. The deformed geometries can include adjunct super-elements or Craig-Bampton substructures. The same methodology has been successfully applied to time history data from a commercial kinematic software package using the basic finite element model geometry from MSC/NASTRAN.



Rendered Finite Element Model of a "MPESS" Payload in the STS Cargo Bay

Introduction

The advent of cost efficient CPU has redefined the technical job task description of today's engineers. Intensive I/O applications such as CAD, and numerically intensive applications such as CFD analyses have found their way onto the desktops of engineers by the means of high-end workstations and Pentium based personal computers.

In the world of finite element analysis, engineers have relied heavily in recent years on the combination of more efficient algorithms and declining CPU cost to assist in the design optimization of structural components. The demand for economical structures, propagated by a more competitive global marketplace, has acted as the primary catalyst for the increased definition of today's finite element models. This revolution in model size and fidelity has also been aided by the use of sophisticated pre-processors coupled with automated mesh generators.

This paper addresses an advanced post processing methodology for viewing the results of MSC/NASTRAN generated displacements. The methods used to generate the displacements are incidental, rather it is the post-processing methodology of these displacements that this paper will focus on. The displacements simply have to be compatible to the global displacement vectors in MSC/NASTRAN. To date, animations have been generated for MSC/NASTRAN normal modes, and transient response analyses. In addition the MSC/NASTRAN generated geometries have been used for the animation of rigid body displacement results. In all these cases the methodology presented in this paper has taken advantage of the existing MSC/NASTRAN generated geometries, material properties and structural component definitions all for the benefit of increased visualization of the analysis results.

The desire for this methodology was driven in part, by the difficulty in viewing large substructured finite element models. An example of this would be a complete satellite FEM. These spacecraft models, such as the X-Ray Timing Experiment spacecraft shown in Figure 1, are often made up of numerous instruments coupled to the basic structure. Since any or all of the coupled instrument models can be large, they lend themselves well to modal substructuring techniques. Modal sub structuring remains the preferred method of modeling in the spacecraft industry. This is due primarily to the extremely large and often numerous substructure models that make up a typical satellite FEM.

Until recently, viewing a slow-motion animation of the displacement results from such models would have only been possible using customized. software on powerful mainframe computers or workstations. The post-processing techniques presented in this paper have two major advantages over such approaches. First, the given MSC/NASTRAN model geometries and their associated element definitions have been fully exploited for an animated, three dimensional rendered image of the finite element model. Second, this technique has been implemented on a 486 based personal computer.

General Methodology

The first step performed for this advanced visualization methodology is to transform the MSC/NASTRAN model geometries to a simplified format. Once the simplified static model geometries are established they are easily manipulated for specified deformed geometries.

After being re-defined to a common coordinate system the geometry of the model(s) and the associated element definitions are transformed from a two dimensional state to a three dimensional state for input to the ART[1] rendering software package. This transformation is performed through a set of codes developed at Swales and Associates. These codes were developed to interpolate the specified element geometries found in the MSC/NASTRAN Bulk Data for input to the rendering code. The rendering code then reads the output data and performs the rendering of the now three dimensional elements.

Once the undeformed geometries of the MSC/NASTRAN model are redefined and rendered any deformed geometries can simply be processed in the same manner and replayed sequentially for a animated visual presentation of the analysis results.

Data Conversion of MSC/NASTRAN Model Geometry

Before the MSC/NASTRAN geometries can be formatted for the rendering software, the data file must first be scanned for geometry. The grid entries and their coordinate system definitions are input to a pre-processor and converted to the basic system. For ease of manipulation they are also ordered sequentially or in MSC/NASTRAN nomenclature, external order. This methodology also addresses the issue of viewing substructures represented by typical modal synthesis methods such as the Craig-Bampton[2] format. A common frustration in the spacecraft industry in regards to modal models is that viewing of analysis results is often difficult due the redundant grid identification between substructures. Perhaps the most distinct advantage of this methodology is the fact that it does not require unique identification of grids and elements across substructures or other physical model boundaries. The renderer itself provides the environment in which substructures can be viewed. Each substructure or physical model need only be transformed to a common viewing system. This transformation can be input manually or if an interface has at least three non-linear points, the transformation will be automatically derived, given the intended connectivity. A substructure model viewed in this fashion can be checked visually for the proper connectivity and orientation.

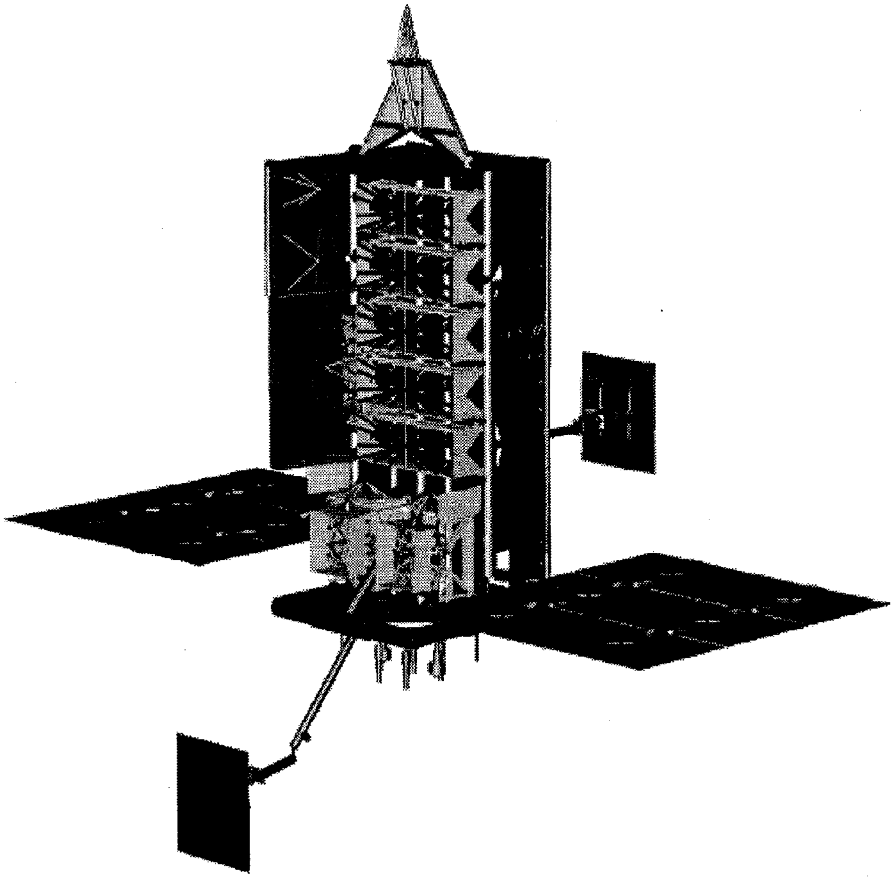


Figure 1

X-Ray Timing Experiment Spacecraft Shown with Ten Sub-Structures

In the case of a completely physical super element model the same methodology can be directly implemented. The MSC/NASTRAN data conversion software has no hard limits and has been used on models containing as many as 60,000 grids. The rendering software package uses dynamic allocation in conjunction with virtual memory and will simply run somewhat slower if physical memory is exceeded. If external super elements have been used then they can only be viewed if the physical Bulk Data files are available. This case would be indistinguishable from the Craig-Bampton methodology discussed above. This feature has proven itself very useful in regards to verifying the orientation and location of multiple substructures coupled to a physical model.

Data Manipulation of Analytical Results

Normal Mode Shapes

For normal mode shapes the displacement vector for each individual model is transformed to system level. A maximum for all substructures is then obtained for each mode and these are used to establish the scale factor for viewing. A value of 20 percent of the maximum structural dimension seems to be reasonable in most instances. This scaled vector can then be used for static deformed plots of the mode shape or as the basis for animation frames.

The number of frames used for the animation will dictate the quality and efficiency of the animation. An odd number of frames is convenient because the center frame is undeformed and need only be generated once for all modes. Seven frames seems to strike a good balance between smooth viewing and computational efficiency. The maximum deflection is then scaled by the cosine of equally divided angles between zero and 180 degrees. For example, to create seven divisions one would take 180 degrees divided by six or 30 degrees. The scale factors would then be 1., .866, .5, 0., -.5, -.866, -1. This sinusoidal scaling generates a smooth and realistic animation when played forward then backward repeatedly.

Transient Response Data

Animated flight transient response data has been applied mainly to the landing transient load cases of the Space Transportation System (STS). It is important to decide what frequency range is of interest for viewing because of associated cost of numerous frames over a wide frequency bandwidth. A typical STS landing transient analysis may yield as many as 2000 time frames. While it is easy for the analyst to obtain response data from a wide range of frequencies, an observer can only really appreciate one or two octaves. It is possible to perceive more than that but it is not useful for diagnostics. For this reason it is convenient to establish a target frequency range for viewing purposes. Frequencies significantly above the selected range will be clipped and those below will be so subtle as to not be noticed.

The example of a Space Shuttle landing transient was observed with shock spectra plots to have the majority of its response between 10 and 40 Hz. With this in mind it was decided that a frame spacing of .006 seconds would be adequate. This interval provided a significant reduction in both the time for rendering and the size of the resulting animation.

The transient data is often scaled for viewing since a typical payload may have dimensions well over one hundred inches with deflections of less than an inch. In this case an unscaled plot would not be very informative. It may flicker a little but wouldn't show what type of movement was occurring. It is possible though to generate unscaled animations of cutaway and zoomed views to observe clearance problems internal to a given structure.

Conversion of MSC/NASTRAN Elements

A set of software routines have been developed at Swales which convert the MSC/NASTRAN bulk data deck geometries for input to the ART rendering code. Complete advantage has been taken of the element geometries and material properties that exist in the Bulk Data information. It is these element properties and material definitions that determine the three dimensional characteristics of the individual elements. For example PBAR and PBEAM entries are read and the values of area and material specifications determine the elements size and color. To enhance depth perception when visualizing the final rendered elements all CBAR, CBEAM and CONROD elements are given a spherical geometry by means of polygonal interpolations. Interpretation of two dimensional plate elements does not account for plate thickness. Depth perception of these elements is often manipulated through light source coordinates in the rendering software. Rigid elements are also represented as tubes but with an arbitrary radius chosen by the user. Concentrated masses can be optionally shown as geometric shapes though this tends to be somewhat confusing if they have been used extensively.

The size of the resulting output files for input to the renderer are not necessarily a function of the number of elements but also the types of elements. The geometry specifications for rendering bar elements, for example are much greater than those of simple plate elements. Currently all MSC/NASTRAN elements are supported for this conversion procedure.

Figure 2 depicts a typical plot of a spacecraft finite element model. Figure 3 shows this same model again after it has been converted and rendered with the methodology discussed above.

The renderings presented here were generated on a 486/66 with 30 megabytes of memory and a 500 megabyte hard drive. The process is computationally intensive so a fast machine plays a role in the overall efficiency. It is advantageous to have a graphics card that supports 26 million colors so that the image can be viewed while it is being created. Systems that can support 256 color graphics will work but the color palette must be reduced prior to rendering.

Assembly and Viewing of Movie Files

The individual picture files generated by the rendering software are assembled using a flic file assembler also developed at Swales. These flic files load the first frame and then compute the difference between subsequent frames. The movie files are then viewed using the Autodesk Animator which is available in the public domain. This viewing software gives complete control over the speed of the play including single frame advance.

Rigid and Flexible Body Dynamic/Kinematic Simulations

The advance visualization methodology discussed above has also been applied to the dynamic/kinematic results of flexible and rigid body simulations. These simulations have been mostly for on-orbit free-free deployments of spacecraft appendages. The development of this visualization technique was prompted by the dynamic simulation of the STEP MISSION II spacecraft after its simulated jettison from the avionics deck of the PEGASUS launch vehicle. Acceleration vectors were applied at the spacecraft to launch vehicle interface in an attempt to account for the forces due to separation. In this case a total of nine rigid bodies participated in the response dynamics of the simulation. As is the case with most rigid body dynamic/kinematic software packages the geometric specifications of the modeled structure are not explicitly required for the analysis but are used mainly for visualizing the results. It was thought that the complicated deployment geometry would be greatly enhanced by the existing detailed geometry of the MSC/NASTRAN FEM. This on-orbit FEM configuration already existed for the normal modes evaluation of the free-free deployed configuration. Clearly a method for visualizing the large displacement geometries by means of the MSC/NASTRAN model would be beneficial, if shown to be efficient.

A simple transformation algorithm was developed to map the displacement output data from the rigid body simulation to the MSC/NASTRAN finite element model for each time increment. These time dependent geometries would then be rendered and played back much in the same way as an animated mode shape or time history transient response.

The transformation shown in Equation 1 (at the end of the paper) was developed for the geometric time history parameters output from the DYNAMIC ANALYSIS DESIGN SYSTEMS (DADS) software package. In this example the Euler symmetric parameters or quaternion components[3] were output for each specified time step at the center of gravity for each rigid body. Equation 1 was then coded for this particular output specification and format. The basic grid point location of the points of connectivity between rigid bodies must be identical between the rigid body model and the MSC/NASTRAN model for an accurate visualization of the simulation results.

This effort has shown that in applications to the spacecraft industry, exploitation of the MSC/NASTRAN model geometries for displaying the time history of large displacements is both economical and efficient.

Conclusions

An advanced methodology for viewing MSC/NASTRAN generated analyses results has been presented. Element definitions and grid point geometries existing in the MSC/NASTRAN Bulk Data entries have been taken advantage of for the purpose of enhanced viewing of undeformed and deformed configurations of the finite element model. This methodology has been extrapolated to the analysis results of a rigid body software package.

Visualization of analysis results is of immense benefit to both engineer and management. The colors and shading of a 3-D realistic rendition have proven to

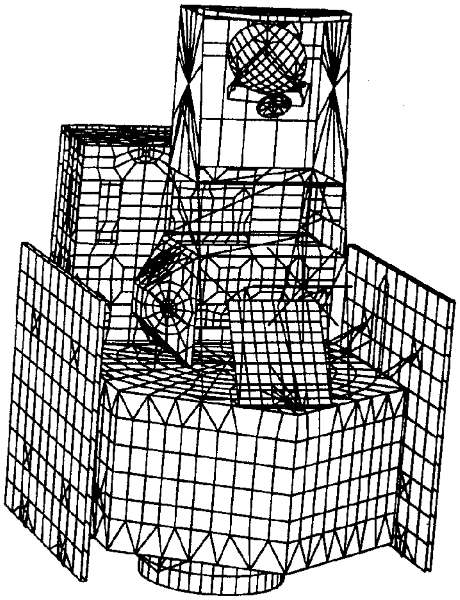


Figure 2
Conventional Finite Element Model Plot
"Tropical Rainfall Measuring Mission" Satellite

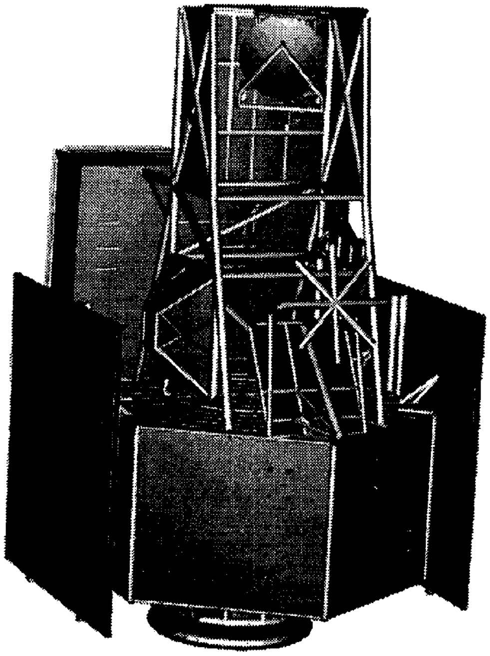


Figure 3
Rendered Image of Finite Element Model
"Tropical Rainfall Measuring Mission" Satellite

be more informative than the typical line plots presently in use. The original intent of this effort, which was merely to provide presentation material, has been overshadowed by the diagnostic benefits to analysts.

Please note that the rendering technology presented in this paper is not new. Similar techniques have been in use for quite some time. What is of note is the application of this type of technology for the visualization of MSC/NASTRAN finite element analysis results.

References

1.) ART V2.0 Users Manual - Analytical Rendering Toolkit Version 2.0, Norton Engineering & Software Technologies, Laural, Maryland

2.) Craig, R.R.,and Bampton, M.C.C., "Coupling of Substructures for Dynamic Analysis, "*AIAA Journal*, Vol.6, No.7, July 1968.

3.) *Spacecraft Attitude Determination and Control*, Edited by: Wertz, J.R., Kluwer Academic Publishers, 1991.

$$\begin{bmatrix} \boldsymbol{X}_{ij}^1 \\ \boldsymbol{X}_{ij}^2 \\ \boldsymbol{X}_{ij}^3 \end{bmatrix} = \begin{bmatrix} \boldsymbol{T}_i^1 \\ \boldsymbol{T}_i^2 \\ \boldsymbol{T}_i^3 \end{bmatrix} + \begin{bmatrix} e_{11_i} & e_{12_i} & e_{13_i} \\ e_{12_i} & e_{22_i} & e_{23_i} \\ e_{13_i} & e_{23_i} & e_{33_i} \end{bmatrix}^T \begin{bmatrix} \varepsilon_j{}^1 \\ \varepsilon_j{}^2 \\ \varepsilon_j{}^3 \end{bmatrix} \quad (1)$$

where

$$e_{11_i} = \phi_{1i}{}^2 - \phi_{2i}{}^2 - \phi_{3i}{}^2 + \phi_{0i}{}^2$$
$$e_{12_i} = 2(\phi_{1i}\phi_{2i} + \phi_{3i}\phi_{0i})$$
$$e_{13_i} = 2(\phi_{1i}\phi_{3i} - \phi_{2i}\phi_{0i})$$
$$e_{21_i} = 2(\phi_{1i}\phi_{2i} - \phi_{3i}\phi_{0i})$$
$$e_{22_i} = (-\phi_{1i}{}^2 + \phi_{2i}{}^2 - \phi_{3i}{}^2 + \phi_{0i}{}^2)$$
$$e_{23_i} = 2(\phi_{2i}\phi_{3i} + \phi_{1i}\phi_{0i})$$
$$e_{31_i} = 2(\phi_{1i}\phi_{3i} + \phi_{2i}\phi_{0i})$$
$$e_{32_i} = 2(\phi_{2i}\phi_{3i} - \phi_{1i}\phi_{0i})$$
$$e_{33_i} = (-\phi_{1i}{}^2 - \phi_{2i}{}^2 + \phi_{3i}{}^2 + \phi_{0i}{}^2)$$

and

$\phi_{1i}, \phi_{2i}, \phi_{3i}, \phi_{0i}$ = Components of the quaterion or the Euler parameters for each time step at the center - of - gravity of the rigid body from the simulation

i = Data output time step from simulation

j = Number of NASTRAN grids for the rigid body

$\boldsymbol{T}$ = Translational response at the C.G. of the rigid body from the simulation

ε = NASTRAN grid point coordinates relative to the equivalent Euler parameter reference point for a given rigid body

$\boldsymbol{X}$ = Transformed NASTRAN grid point geometry for the each translational direction