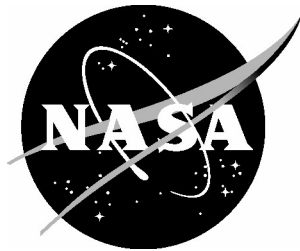NASA/TM-2006-214500

Stochastic Simulation Tool for Aerospace Structural Analysis

Norman F. Knight, Jr.
General Dynamics – Advanced Information Systems, Chantilly, Virginia

David F. Moore
Langley Research Center, Hampton, Virginia

August 2006

The NASA STI Program Office . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the lead center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:
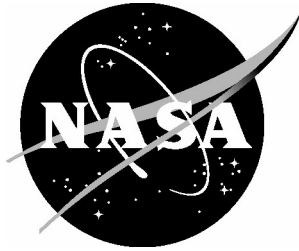
- TECHNICAL PUBLICATION. Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counterpart of peer-reviewed formal professional papers, but having less stringent limitations on manuscript length and extent of graphic presentations.

- TECHNICAL MEMORANDUM. Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.

- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.

- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or co-sponsored by NASA.

- SPECIAL PUBLICATION. Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.

- TECHNICAL TRANSLATION. English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized databases, organizing and publishing research results ... even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at *http://www.sti.nasa.gov*

- E-mail your question via the Internet to help@sti.nasa.gov

- Fax your question to the NASA STI Help Desk at (301) 621-0134

- Phone the NASA STI Help Desk at (301) 621-0390

- Write to:
  NASA STI Help Desk
  NASA Center for AeroSpace Information
  7121 Standard Drive
  Hanover, MD 21076-1320

Stochastic Simulation Tool
for Aerospace Structural Analysis

Norman F. Knight, Jr.
General Dynamics – Advanced Information Systems, Chantilly, Virginia

David F. Moore
Langley Research Center, Hampton, Virginia

Available from:

Stochastic Simulation Tool
for Aerospace Structural Analysis

Norman F. Knight, Jr.
General Dynamics – Advanced Information Systems
Chantilly, VA

David F. Moore
NASA Langley Research Center
Hampton, VA

Abstract

Stochastic simulation refers to incorporating the effects of design tolerances and uncertainties into the design analysis model and then determining their influence on the design. A high-level evaluation of one such stochastic simulation tool, the MSC.Robust Design[1] tool by MSC.Software Corporation, has been conducted. This stochastic simulation tool provides structural analysts with a tool to interrogate their structural design based on their mathematical description of the design problem using finite element analysis methods. This tool leverages the analyst's prior investment in finite element model development of a particular design. The original finite element model is treated as the baseline structural analysis model for the stochastic simulations that are to be performed. A Monte Carlo approach is used by MSC.Robust Design to determine the effects of scatter in design input variables on response output parameters. The tool was not designed to provide a probabilistic assessment, but to assist engineers in understanding cause and effect. It is driven by a graphical-user interface and retains the "engineer-in-the-loop" strategy for design evaluation and improvement. The application problem for the evaluation is chosen to be a two-dimensional shell finite element model of a Space Shuttle wing leading-edge panel under re-entry aerodynamic loading. MSC.Robust Design adds value to the analysis effort

---

[1] MSC is a trademark of MSC.Software Corporation.

by rapidly being able to identify design input variables whose variability causes the most influence in response output parameters.

Introduction

Engineering design has evolved over the past decades as analysis tools and computing systems evolved. The engineering design process has changed from merely determining a specific set of values for the design variables that meet certain design objectives (deterministic design process) to an assessment of design sensitivities that account for tolerances and uncertainty. This trend in engineering design results in the quantification of risk (risk-based design process). Originally this design process was performed manually by relying on the intuition of the designer to explore the design space. A single-point design is analyzed and perhaps a few parametric studies are evaluated. Schedule and cost generally precluded any thorough evaluation of the structural analysis model to understand and assess how uncertainty may influence the design's success (or failure). Uncertainty is usually accounted for through the use of safety factors, and the trend in structural design is to rely more heavily on the analysis results to fill in the gaps where test data are not available. Later, design optimization procedures using gradient- and non-gradient-based algorithms were coupled with engineering analysis tools and became available on high-performance computing systems. This evolution automated the previous manual design iteration process and permitted a more thorough search of the design space. However, only recently have the influences of uncertainties due to manufacturing tolerances or material variability been accounted for in the engineering design analysis.

Probabilistic design methodologies [1-4] that consider variability and tolerances in the design variables, rather than fixed or mean values alone, are becoming available and are migrating into the engineering design process. In essence, the point-design process is morphing to a process leading to design success by accounting for tolerances and uncertainty. As a result, the reliability of the design can be determined through statistical methods. This type of engineering design approach is referred to as reliability-based design or risk-based design. To support such a design approach, analysts need analysis tools to assist in establishing the credibility, accuracy, and robustness of the simulation and to identify potential problems associated with design variability early in the design process in order to guide the design process and to anticipate and reduce the testing requirements (*i.e.,* better tests).

The impetus to reduce testing and prototype costs during the design cycle has resulted in an increased reliance on engineering analysis models. This reliance places an ever-increasing responsibility on commercial software vendors as well as on engineering analysts to ensure the correctness and applicability of their analysis tools and models to simulate the design performance. Computer systems and engineering analysis software, such as MSC.Nastran[2] [5], are now so powerful and so readily available that it is common place for analysts and project stakeholders to accept large complicated finite element models as accurate. Often the acceptance of a finite element model as accurate is based on perception and is not warranted. A finite element model is perceived to be accurate and correct if: (1) it involves a large number of nodes and finite elements to represent the design, (2) the geometry is represented accurately through the use of solid geometry definitions, (3) the analysis results can be visualized and animated (immersive technology may even be integrated with the analysis tool), and (4) the solutions required a significant amount of computational resources. The analysis tool itself may be verified and validated. However, each application of the analysis tool to a new design problem requires the development of a new engineering analysis model that needs to be assessed in order to establish the accuracy of the new model – not the tool. Therefore analysts need to establish the credibility of their analysis model, verify the results through correlations with test data, and calibrate the simulation for variability in the design. In addition, stakeholders need to be vigilant and require proof that the computational simulations obtained using a given analysis tool and a given engineering analysis model are in fact credible solutions.

The question then is how do analysts establish the credibility of the engineering analysis model. A two-step process related to accuracy and robustness is suggested. Accuracy refers to the ability of the analysis model to capture correctly the perceived important physical behavior of the design. Robustness refers to understanding the sensitivities of the analysis model to design parameter variability and problem uncertainties. These two steps contribute to establishing credibility in the computational simulation results.

First, for an engineering analysis model to be accurate, it must represent the engineering science and mechanics of the design problem. In a formal sense, engineering analysis models based on

---

[2] MSC.Nastran is a trademark of MSC.Software Corporation. NASTRAN is a registered trademark of NASA.

fundamental mechanics principles (conservation laws, equilibrium principles) and cast in a computational framework are considered physics-based engineering analysis models. In an applied or working sense, engineering analysis models that (1) represent the physical geometry, boundary and initial conditions, and constraints; (2) replicate the constitutive behavior; (3) correlate with experimental data; and (4) provide credible predictive solutions within a bounded range of interest and within bounds of known data are defined as physics-based engineering analysis models. Characteristics of physics-based engineering analysis models include a formulation based on basic principles of mechanics; the use of constitutive models correlated with test results over the range of interest; the use of a proven computational framework or analysis tool; an accurate representation of the physical problem (*e.g.,* geometry, generalized imperfections, boundary and initial conditions including load introduction, constitutive models); working within established bounds and limits of applicability; and the generation of credible, consistent, and reliable predictions correlated with globally and locally observed experimental evidence. Second, for an analysis to be robust, it needs to ensure that sensitivity of design performance metrics to tolerances and uncertainties are defined, understood, and mitigated, if possible. These two steps have the potential to impact engineering design analysis in a positive way.

MSC.Robust Design (or simply RD) [6,7] is a stochastic simulation tool that contributes to increasing the understanding of how uncertainty in a finite element analysis model may affect the design by performing stochastic simulations in a near-automatic fashion. These stochastic simulations are based on a Monte Carlo approach that automatically generates a user-specified number of unique, independent combinations of the input variables. In essence, a new analysis model is derived from the baseline analysis model for each combination of design input variables. The collection of these combinations of design input variables and their corresponding response output parameters (observables) defines a "meta model" for the design based on a given finite element model. RD submits an analysis task for each combination of the design input variables (*i.e.,* generates a new `*.bdf` file) to MSC.Nastran [5] either serially or concurrently depending on the available computer system and software license agreements. For conducting the MSC.Nastran runs concurrently, RD submits the `*.bdf` files to the MSC.Analysis Manager for distribution to the computer systems with MSC.Nastran available. The output performance metrics (or observables) from each finite element analysis are then implanted into the RD

database for subsequent post-processing. RD post-processing through a graphical-user interface (GUI) is performed by the analyst to access the resulting meta-model. A meta-model contains large amounts of information with interrelationships between input design variables and observable output parameters. The meta-model can be viewed by looking at any combination of two or three variables (*i.e.,* two- and three-dimensional plots). RD uses a *decision map* to provide a means to view the meta-model in a manner that can be quickly understood. A *decision map*, described more in subsequent sections, shows the correlations between input design variables and observable output parameters.

The objective of the present paper is to summarize a high-level evaluation of the MSC.Robust Design stochastic simulation tool. Various aspects of the stochastic simulation tool are described and illustrated through the use of an aerospace structural analysis problem of current interest. While other application problems were also studied during this evaluation, the evaluation reported here focuses on an application problem from the Space Shuttle program and the Return-to-Flight activities. The evaluation is performed using the historical two-dimensional shell finite element model of a Space Shuttle wing leading-edge (WLE) panel subjected to re-entry aerodynamic loading. Background information on the application problem is provided in Refs. 8-15. An outline of the present paper is as follows. First, the present paper briefly describes the application problem. Then features of the RD software are described using the application problem as an illustrative example. Next, results and observations from the application problem are discussed. The present paper closes with a summary.

Application Problem

The Space Shuttle Orbiter's WLE panels provide the aerodynamic load-bearing capability and thermal-control capability for areas that exceed 2,300-degrees Fahrenheit [8-15]. There are 22 panels of reinforced carbon-carbon (RCC) along the leading edge of each wing of the Space Shuttle Orbiter as indicated in Figure 1. A typical RCC WLE panel assembly is shown in Figure 2. Because the shape of the wing changes from inboard to outboard, each RCC WLE panel is unique. RCC WLE panel thickness values range from 0.233-to-0.512-inches thick (*i.e.*, 19 plies to 38 plies) depending on the location on the panel and the location along the wing (see Ref. 9).

The historical, certified design finite element analysis models of the RCC WLE panel assemblies are defined as the set of MSC.Nastran finite element models developed during the early stages of

the Space Shuttle program (1970's time frame). The historical, certified finite element model of RCC WLE Panel 4 assembly is shown in Figure 3. Not every WLE panel on the Space Shuttle Orbiter wing is modeled explicitly because several are geometrically similar. Panels exposed to the highest loading during ascent and re-entry are analyzed. The RCC WLE panel structural models are two-dimensional shell finite element models with selected airload pressure cases defined for the various panels. They are considered to be the "flight certified" models for loads and strength because the geometry, material process and properties, along with the metal clevis and interfaces including field-break mechanisms were verified for Space Shuttle missions. These analysis models were also test verified for air loads, temperature and mass loss per design requirement.[3]

In these MSC.Nastran finite element models, the RCC WLE panel and adjacent T-seal (blue components in Figure 2) are modeled using two-dimensional shell finite elements (see Figure 3) with membrane and bending stiffness terms specified directly rather than implicitly through the usual engineering mechanical properties (*i.e.,* elastic modulus and Poisson's ratio). These stiffness terms are "apparent" values based on the overall laminate dimensions, and hence the RCC material is assumed to be a linear elastic, homogeneous, transversely isotropic material, which is consistent with the available material data. Apparent engineering material properties for different RCC laminate thickness values are given in Ref. 14. As a result, the elastic moduli for tension, compression and bending are independently defined for different thickness values and also as a function of temperature based on the material data in Ref. 14. RCC WLE panel thickness variations occur throughout the panel with thicker regions located near the edges. Material stiffness coefficients for membrane (tension and compression) and for bending are defined for various RCC thickness values within the RCC WLE panel finite element models.

Input of the RCC material data to MSC.Nastran is through the MAT2, MATT2, and TABLEM1 records of the bulk data file (`*.bdf`). Interested readers should refer to Ref. 5 for details on these input records. MAT2 input records define the room-temperature values of the stiffness coefficients G_{ij} according to the equation:

---

[3] Based on personal communication with T. Ho, Lockheed-Martin, Dallas, TX.

$$\begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} G_{11} & G_{12} & G_{13} \\ G_{12} & G_{22} & G_{23} \\ G_{13} & G_{23} & G_{33} \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \gamma_{12} \end{Bmatrix} - (T - T_{ref}) \begin{Bmatrix} A_1 \\ A_2 \\ A_{12} \end{Bmatrix} \right)$$

or

$$\{\sigma(T)\} = [G(T)]\left(\{\varepsilon\} - (T - T_{ref})\{A(T)\}\right)$$

MATT2 input records identify table numbers for those parameters exhibiting temperature dependence, and the TABLEM1 input records define the variation of a given stiffness coefficient as a function of temperature. The stiffness coefficients G_{ij} for the original MSC.Nastran finite element models of the RCC panels are defined as:

$$G_{11} = \frac{E}{1 - v^2} = G_{22}$$
$$G_{12} = vG_{11}$$
$$G_{33} = G$$
$$G_{13} = G_{23} = 0$$

where E is the elastic modulus (tensile or compressive), G is the shear modulus, and v is Poisson's ratio. Values for the apparent elastic modulus are defined in Ref. 14 for tension, compression, and bending for different RCC thickness values and as a function of temperature. The A_1, A_2, and A_{12} coefficients define the coefficients of thermal expansion where A_{12} is taken as zero. The reference temperature T_{ref} is taken as room temperature. No temperature effects are included in the analyses reported in this paper (*i.e.*, $T = T_{ref}$).

These finite element models are rather coarsely discretized for stress analysis and damage assessment.[4] However, some trends may be determined by using these analysis models, and those trends should be confirmed using the more contemporary detailed finite element models developed under Return-to-Flight activities. In the analysis models for these studies, the RCC material is defined using the MAT2 input records in terms of material stiffness coefficients G_{ij}. Each load case is solved perhaps several times, and the material property assignments (tensile or compressive) are adjusted so that the shell elements with membrane compression stress use

---

[4] These finite element models were developed in the early 1970's, and at that time, they were probably considered to be state-of-the-art finite element models that challenged the existing computational environment.

compression property data and shell elements with membrane tension stress use tension property data.

The wing leading-edge front spar and its attachment hardware (non-blue components in Figure 2) are modeled with equivalent stiffness terms defined using the DMIG input records at four attachment points on the RCC panel finite element model. The air loads are defined using the PLOAD2 input records for distributed surface loading. The T-seal is "attached" to the panel through a set of multi-point constraints (MPCs) and linear elastic spring elements (CELAS2). This attachment modeling is dependent on the load case considered, and the bearing surface between the T-seal flanges and the RCC panel edges also changes when the air loads change. The geometry for each wing leading-edge RCC panel is different thereby requiring finite element models for several panel assemblies.

The historical MSC.Nastran finite element model for RCC WLE Panel 4 shown in Figure 3 has 1,279 nodes, 41 CBAR elements, 130 CELAS2 elements, 34 MPC constraints, 1,067 CQUAD4 elements and 132 CTRIA3 elements for the panel and T-seal combination. This finite element model is used extensively in Ref. 15. It is representative of a general finite element analysis model involving different finite element types, different material definitions (*i.e.,* engineering mechanical properties and stiffness coefficients), different element properties definitions (*i.e.,* shell element thicknesses and beam cross-sectional properties), both single- and multi-point constraints, and a distributed surface loading.

Description of the MSC.Robust Design Tool

MSC.Robust Design (RD) software tool [6,7] is a recent product of MSC.Software Corporation for assessing the solution variability of a given finite element analysis model to various random input design variables. These design variables may include engineering material properties (individually or as a set), plate/shell thickness values, beam cross-sectional properties, and applied load magnitudes. The structural response of the finite element model to random input of these design variables is then determined based on user-selected output response metrics such as a maximum resultant displacement, maximum or minimum stress component, lowest vibration frequency, or smallest buckling load. The input variables and output response metrics (or observables) are currently limited (*i.e.,* using RD Version 2004r3) to quantities directly accessible in the MSC.Nastran `*.bdf` input file and MSC.Nastran `*.f06` output file.

RD is based on a graphical-user interface (GUI) for pre- and post-processing. The primary RD display window shown in Figure 4 consists of three partitions: the project tree (along the left side of the window), the design table (along the bottom edge of the window), and the graphical view port (remaining part of the window). Each partition can be sized independently. As with most GUI tools, a number of toolbars and pull-down menus are available (see Ref. 6). RD has two primary functions. One function is to perform stochastic simulations, and the other function is to perform stochastic design improvements. Only the former function is evaluated in the present paper.

RD does not assess the adequacy of the finite element model itself, but rather assumes that the imported finite element model has been verified by the analyst as an appropriate and credible engineering analysis model for the analysis task, as consistent with the analysis objectives, and produces credible solutions for the nominal baseline design as it is understood. The analyst continues to bear the responsibility to assure that the choice of finite element mesh, selection of finite element types, selection of material models, imposition of boundary conditions, definition of coordinate systems, and application of loads are accurately and correctly defined. RD examines the design based on a given baseline finite element model and determines the response variability to each of the input design variables.

RD begins by reading the MSC.Nastran bulk data file (`*.bdf`) and automatically generates a "project tree" similar to the one shown in Figure 5. The bulk data file for RCC WLE Panel 4 was read, and the RD project tree was generated based on the MSC.Nastran bulk data file input record types. For example, in Figure 5, the different finite element types found in the `*.bdf` file generate different branches on the project tree along with cascading branches for their associated material and property definitions. Table input, loads, constraints, and other data from the `*.bdf` file are also indicated in the project tree. In addition, the output requests by the analysis subcase are identified. In this example, only a linear stress analysis is performed (Solution Sequence 101 or SOL101 in MSC.Nastran [5]). RD also supports the stochastic simulation of other MSC.Nastran solution sequences including SOL103 (eigenvalues analysis), SOL105 (linear buckling analysis), SOL108/111 (frequency response analysis, direct and modal), and SOL600 (implicit nonlinear). Finally, the results from executing RD are available under the "Results" branch of the project tree shown at the bottom of Figure 5.

Once RD reads the `*.bdf` file, the input finite element model and the project tree are displayed by RD in the graphical view port shown in Figure 4. This display tool allows the analyst to view the finite element, rotate it, and zoom in or out on the finite element model as a whole. As the analyst selects a particular branch on the project tree (left side of the screen), the corresponding part of the finite element model is highlighted in the graphical view port. Analysts can readily interrogate the finite element model without any prior knowledge of the analysis model. Unlike using MSC.Patran[5] to interrogate the finite element model, the RD GUI can be used to determine material property layout or thickness distributions by simply selecting the appropriate branch of the project tree (*e.g.,* the analyst does not need to define groups of elements by property identification). While RD does provide some basic graphical display tools for viewing the finite element model, it has limited functionality and is neither a replacement for nor an alternative to using MSC.Patran for finite element modeling and post-processing of analysis results (*e.g.,* deformed geometry plots or contour plots of results).

After RD develops the project tree, the analyst may then proceed to define the random variables for the simulation. The random variables can be defined in three ways. A single variable on an MSC.Nastran record can be defined as a random variable, all entries on an MSC.Nastran record can be defined as random variables, or all records in a select group can be defined as random variables. These choices are made easily through the RD GUI by simply identifying the variable(s) on the project tree and assigning their variation a distribution (*e.g.,* Gaussian, uniform, Weibull, logarithmic, or even a discrete distribution) and setting a coefficient of variation. The coefficient of variation (CoV) is defined as the ratio of the standard deviation to the mean (or nominal) value multiplied by 100%. The specific value in the MSC.Nastran bulk data file for each variable is considered as the mean or nominal value, and the range of values for each input design variable is then determined automatically by the original input value, the CoV, and the cutoff value in terms of the number of standard deviations. These choices are displayed in the design table at the bottom of the display window under the Simulation Input tab as shown along the bottom of Figure 4. For example, if the analyst chooses to randomize the elastic modulus of a material type (see Figure 6a), then the type of random distribution needs to chosen, the cutoff value in terms of the number of standard deviations needs to be defined, and the

[5] MSC.Patran is a trademark of MSC.Software Corporation.

coefficient of variation needs to be specified (see Figure 6b). Each distribution type has default values set for the cutoff and coefficient of variation that can be modified by the user. The default values for a Gaussian distribution are a coefficient of variation of 5% and a cutoff of three standard deviations.

The analyst selects the output response variables to be considered from those available in the MSC.Nastran output file (*.f06 file) in a similar way. Options include selecting minimum and/or maximum values of the stresses either by component direction, principal values, or von Mises value as shown on the project tree partition of Figure 4. Displacements, rotations, constraint forces, and eigenvalues (vibration frequencies and buckling factors) are also available depending on the solution sequence being executed. In this example, only the linear elastic stress analysis (SOL101) is performed.

In addition, the analyst needs to specify the number of stochastic simulations (or number of samples) to perform as indicated in Figure 7. RD is based on a Monte Carlo approach and automatically generates a set of unique analysis models based on the uncertainty variables defined for the problem using an updated Latin hypercube sampling technique. RD performs the number of complete MSC.Nastran simulations specified by this number-of-samples value plus one (*i.e.,* 100 solutions in addition to the solution of the imported baseline finite element model). It is important to realize that each analysis represents a unique combination of the random input design variables – no combination sample is replicated. Collectively the results for each combination define the meta-model for RD post-processing. RD generates new MSC.Nastran bulk data files for each combination of the randomized variables, and each combination is then submitted to MSC.Nastran to be solved. An option for saving each *.bdf file is also available but is not the default.

RD submits these analysis models for solution using either a single computer or multiple computers that are networked or clustered together. As each deterministic simulation based on each set of random input variables is performed, the results for the output metrics, not the entire MSC.Nastran analysis database, are automatically archived in the RD database. A typical summary of archived results in the RD database is illustrated in Figure 8. In this view, the project tree is shown on the left side of Figure 8 in its "collapsed state" whereas its "expanded state" is shown in Figure 4. Along the bottom portion of Figure 8, some of the observable results

parameters are shown. Different displacement results (maximum and minimum values for the resultant magnitudes and by component direction) are displayed in the design table portion of the display window. Each output parameter has its minimum, maximum, mean, and most-likely values given as well as the coefficient of variation.

After these independent analyses are completed and archived within the RD database, the solution set can be studied to examine the scatter in the output results metrics (observables) based on the scatter in the input design variables. This collection of input and output values is defined by RD as the "meta-model" of the system, which is graphically displayed as a decision map (see Figures 9 and 15). Large amounts of information are contained in these interrelationships, which are shown as links between design input variables and response parameters in terms of correlation factors [6]. For each combination of stochastic variables, RD computes a linear correlation coefficient (Pearson coefficient) and a nonlinear correlation coefficient (Spearman-rank coefficient) based on the specific stochastic variables and their mean values. These correlation coefficients range from ±1. Values close to ±1 indicate strong correlation, and values close to zero indicate very weak or no correlation. Correlation is similar to a sensitivity analysis. In a sensitivity analysis, the effect the variation of an input variable has on an output function is determined while generally holding all other variables constant. A correlation is done the same way, but with all of the other variables changing within their defined ranges.

Within RD, the meta-model data are presented to the analyst using three different display tools. The first display tool is a *decision map*, such as the one shown in Figure 9a. A decision map is a graphical structuring of the information indicating the degree of interaction between input variables and output parameters. Across the top of the display is information on the number of random design input variables, the number of response output parameters, the number of correlations greater than a specified value, and the effective minimum correlation factor. Along a diagonal line in the display view port, one or more input variables (yellow or orange) followed by one or more output variables (red or blue) are displayed. Interactions between input and output variables (referred to as a link) are indicated by the lines connecting individual variables (see Figure 9a). As the cursor is placed on a diagonal symbol, the definition of that symbol is displayed. Alternatively, the labels for all variable names can be activated as shown in Figure 9b; however, this view tends to crowd the decision map.

The second display tool is the *scatter plot*, which is driven by selections made by the analyst using the decision map (like the one shown in Figure 9a). This scatter plot is also referred to as a "cloud" or a "two-dimensional ant-hill plot". The analyst can view a scatter plot either by selecting a pair of interacting variables and choosing the scatter-plot-display option from the RD toolbar, or by clicking on the node on the decision map connecting the variables. Then, the distribution of all deterministic solutions from the stochastic simulations (*i.e.,* the 101 solutions in this case) are displayed as an *xy*-plot in Figure 10 for the selected pair of interacting input and output variables from the decision map shown in Figure 9. A few of the combinations of random input design variables result in "outliers" (see Figure 10); that is, responses that are away from the majority of the solutions or on the edges of the cloud. Understanding why certain combinations, within the bounds of the design variables, generate solutions on the fringes contributes to design improvement.

In addition, a three-dimensional scatter or ant-hill plot as shown in Figure 11 can be generated by selecting a combination of three input and output variables. This three-dimensional plot may be rotated and examined in different ways to determine if the combination set has solutions lying close to a plane, if the combination is truly three dimensional, or if the combination contains clusters or groupings of results. These three-dimensional plots often expose non-intuitive interactions that may potentially impact the design performance and functionality.

The third display tool is the *pie-chart display* as shown in Figure 12, which is driven by a response output parameter selection made by the analyst from the decision map. The pie-chart display gives an indication of how the scatter in all design input variables influences the scatter of a specific response output parameter. The design input variables having the most influence on the scatter of the selected response output parameter are readily detected from the pie-chart display (*i.e.,* those variables representing the larger pieces of pie). A representative pie-chart display is shown in Figure 12. Nine variables having the larger influence are presented separately, while the remaining input parameters are lumped together and labeled "Others" on the display. Ideally, the scatter in the structural response should not be strongly dependent on any one, or even on a few, design input variables, and so a pie-chart display, like that shown in Figure 12, is desirable. However, if one or more design input variables do strongly influence the scatter of the structural response, a pie-chart display will reveal that dependency. The pie-chart

display provides the analyst a tool to aid in understanding the importance of design variable tolerances and uncertainty.

Results and Observations

In the preceding sections, the MSC.Robust Design stochastic simulation tool and the application problem used in this evaluation are briefly described. Finite element linear stress analyses are performed using a desktop personal computer with a 3.19-GHz Xeon processor and 3 GB of memory and running Windows XP, MSC.Nastran Version 2005r2 [5], and MSC.Robust Design Version 2004r3.6 [6]. The original two-dimensional shell finite element model of RCC WLE Panel 4 assembly is selected as a representative aerospace structural analysis problem to use in the high-level assessment of MSC.Robust Design (or RD). The RCC WLE Panel 4 finite element model, shown in Figure 3, has approximately 1,300 nodes and 1,200 two-dimensional shell elements. Each MSC.Nastran linear stress analysis of this finite element model requires approximately 10 CPU seconds to execute on the desktop personal computer being used.

The linear stress analysis of the baseline finite element model of RCC WLE Panel 4 assembly was performed using MSC.Nastran. The loading case corresponds to a re-entry aerodynamic load case where an outward-directed pressure (burst pressure) is applied on the upper panel surface and an inward-directed pressure (crush pressure) is applied on the lower panel surface. The contour plot of the resultant displacement magnitude is shown in Figure 13. The distribution shows the outward deflections on the upper panel surface. Maximum resultant displacement magnitudes occur near the center of the upper panel surface. Contour plots of the von Mises stress distribution for Panel 4 and T-seal 5 on the outer surface of the shell finite elements (Z1 position through the thickness) are shown in Figures 14a and 14b, respectively. The maximum value occurs at the ribs of the panel. Additional results based on this finite element model are reported in Ref. 15.

This finite element model, as defined in the MSC.Nastran bulk data file, was created during the Space Shuttle development period and still was readily read into RD without any difficulties. Input random variables were defined as the RCC thickness variation over the panel, the elastic mechanical properties of the RCC (in terms of the G_{ij} stiffness coefficients on the MAT2 input record), the load amplitude of the aerodynamic pressure (not the spatial distribution of the aerodynamic pressure), the beam cross-sectional properties representing the spanner beams, and

the spring stiffness terms representing various RCC WLE panel attachment hardware. The finite element model, shown in Figure 3 from MSC.Patran and shown in Figure 4 in the graphical view port of the RD display window, represents a single RCC WLE panel assembly. A panel assembly includes the RCC WLE panel, its T-seal, the panel and T-seal attachment hardware, and spanner beams. The attachment hardware that connects the panel assembly to the WLE front spar is referred to as the spar attachment hardware. This hardware and the WLE front spar itself serve to provide the backup structural support for the RCC WLE panel assemblies. In the finite element model, the resulting elastic structural support is provided by a set of stiffness coefficients (DMIG terms) defined at the field-break points on the RCC WLE panel assembly. The DMIG terms representing the stiffness of the backup structure could not be selected as random variables within this version of RD (Version 2004r3). As a result, a total of 635 random design input variables were defined. Although not required by RD, each random input variable was assumed to have a Gaussian (normal) distribution with a coefficient of variation (CoV) equal to five percent and a cutoff range set by three standard deviations (default settings). Potentially each random variable can have a different statistical distribution type, mean, and CoV.

Response output parameters (or observables) were selected from the RD project tree (see Figure 4). The maximum and minimum values of the von Mises stress, the principal stresses, and the stress components (normal and shear) at the element centroids were chosen. The maximum and minimum values of the nodal displacements (by component direction and resultant magnitude) were also chosen as response output parameters. A total of 28 response output parameters resulted; however, six of the 28 parameters are associated with the transverse stress components, which are neglected in these two-dimensional shell finite element analyses.

MSC.Robust Design then computed 101 distinct solutions (by default), which required approximately twenty minutes of wall-clock time to complete using the desktop personal computer described previously. One solution corresponds to the baseline finite element model of the imported MSC.Nastran bulk data input file, while the other 100 solutions correspond to unique permutations of the design input variables (Latin hypercube sampling) as generated by RD automatically. The stochastic simulation results were then graphically displayed in three interrelated forms: decision maps, clouds, and pie charts. *Decision maps* give an indication of the interrelationship of the input variables and the output metrics. *Clouds* display the distribution of these 101 solutions, which can be used to aid in identifying outliers in the design or

sensitivities to change. *Pie charts* give a ranking of all input variables to the scatter in a single output parameter and provide a tool to identify input variables that dominate the scatter of an output parameter.

A representative decision map for the RCC WLE Panel 4 analysis is shown in Figure 9 for a specified correlation factor r_0 of 0.50. Seven dominant design input variables are identified in Figure 9b for the re-entry aerodynamic load case considered. These seven input variables, as described in Figure 9b, are associated with the elastic modulus of the beam elements (Material 100), the beam cross-section properties (Bar 200), 19-ply-thick-tensile-regions thickness (Shell 19), 19-ply-thick-compressive-regions thickness (Shell 119), 22-ply-thick-compressive-regions thickness (Shell 122), T-seal 20-ply-thick-compressive-regions thickness (Shell 220), and the 19-ply-compressive G_{22} stiffness coefficient (Material 119). Twenty-one response output parameters are also identified in Figure 9b, and the interactions between the input variables and output parameters are shown by the various links. The links between the seven input variables and the twenty-one output parameters all have a correlation factor r equal to or greater than the specified value r_0 of 0.50. Links displayed above the diagonal have a correlation factor r larger than $(1 + r_0)/2$ (*i.e.,* larger than 0.75 in this case), and those below the diagonal have a correlation factor r smaller than $(1 + r_0)/2$.

Specifying the correlation factor r_0 on the RD toolbar, changes the amount of information displayed on the decision map. This effect is illustrated by comparing the decision map shown in Figure 9b with the decision maps shown in Figure 15. The decision map shown in Figure 9b is generated using a specified correlation factor r_0 of 0.50 and has an effective minimum correlation factor of 0.507. The effect of changing the specified correlation factor r_0 from 0.50 to 0.75 and then to 0.25 is illustrated in Figures 15a and 15b, respectively. Increasing the specified correlation factor r_0 leads to fewer interactions (or links) being displayed in the decision map as indicated in Figure 15a. Decreasing the specified correlation factor r_0 leads to an increase in the number of interactions (perhaps all) between the input random design input variables and the response output variables as indicated in Figure 15b. As the specified correlation factor is decreased, the relative significance of the decision map links becomes less apparent because the relative significance of the interaction between input and output variables is blurred. However, the use of a small specified correlation factor does clearly indicate the

number of possible interactions between input and output variables. Changing the specified correlation factor helps the analyst identify "tall poles in the tent" quickly and guides the analyst in examining the design to determine the cause of these "tall poles" and finding alternatives that lead to design improvement. These alternative designs may require changes to the baseline finite element model and the need to validate the new analysis model.

By selecting the "Shell 19" input variable and the "Displacement Max Magnitude" response output parameter on the decision map (see Figure 9 or 15), the scatter plot or "cloud" shown in Figure 10 is obtained. Again, this view shows the project tree on the left side, a subset of the simulation output parameters along the bottom in the design table, and the scatter plot for the chosen variables in the graphical view port. The range of the input variable (19-ply-thick-tensile-regions thickness) used in the stochastic analysis is displayed (0.204 – 0.262 inches based on values used in the stochastic analysis and bounded by the user-specified input values – mean value, cutoff range, and CoV). The computed bounds of the response output parameter (Displacement Max Magnitude) from the stochastic analysis are shown on the vertical axis (0.0778 – 0.0967 inches). Every dot in the "cloud" represents one of the 101 deterministic solutions generated for each combination of random input design variables by the Monte Carlo analysis used by the RD software.

A view of this "cloud" by itself is given in Figure 16. The basic trend of the results indicates a reduction in displacement as the thickness increases, which is intuitive. As a result, both the linear and nonlinear correlation factors are negative. If the correlation was in fact linear, then the two correlation factors would be equal. The closer these correlation factors are to ±1, the narrower the "cloud" will be and the stronger the correlation between the input and output variables. The most-likely value is also identified on the cloud by the "+" symbol. This value is defined as the centroid of the bounding polygon of the "cloud" of the solutions. Outlier solutions from the stochastic simulation are also readily identified on the scatter plot as evident in Figure 16.

By selecting "Material 100-E" as the input variable and "Stress Max Principal C" as the output parameter on the decision map shown in Figure 9a, the scatter plot or cloud shown in Figure 17 is obtained. Again, the range of each input and output parameter is indicated on the plot. In this case, the basic trend of the results indicates an increase in stress as the material modulus

increases, which again is intuitive. Hence, both correlation factors are positive. However, for this pair of variables, significantly more scatter is evident in the plot, and the correlation factors are smaller than those observed for the case shown in Figure 16.

Representative pie-chart displays are shown in Figure 18. The scatter in the maximum von Mises stress (as a response output parameter) is influenced more by scatter in the nine design input variables listed in Figure 18a. The most influential design input variable on the maximum von Mises stress is "Shell 19". This variable is associated with the thickness of the WLE panel region of the finite element model having a 19-ply RCC laminate. However, less than 2% of the scatter in the maximum von Mises stress is attributed to the "Shell 19" design input variable. Nearly 94% of the scatter is attributed to the remaining 626 design input variables with each contributing less than 0.5%. Similarly, the scatter in the maximum displacement magnitude (as a response output parameter) is influenced more by the scatter in the nine design input variables listed in Figure 18b. Again, the most influential design input variable on the maximum displacement magnitude is "Shell 19". However, less than 2% of the scatter is influenced by the "Shell 19" design input variable. Nearly 94% of the scatter is attributed to the remaining 626 design input variables.

Applying stochastic simulation tools to debris impact assessment on the Space Shuttle WLE panels is another possible application. Additional design variables would then include the discrete loading event caused by potential debris impact of the WLE panels. Random input variables would include debris velocity, debris mass, debris shape and orientation, and impact location on the WLE panel. Output response parameters could be peak stress, threshold damage indicator, or some other performance metrics. Integration of RD with other engineering analysis tools would need to be developed to accommodate such an evaluation.

Summary

MSC.Robust Design (RD) is a GUI-driven stochastic simulation tool to examine existing MSC.Nastran finite element models and to assess response sensitivities to finite element model input design parameter variability and uncertainty. Design input variations due to uncertainties (*e.g.,* loads), material property variations, or manufacturing tolerances can be considered. Random variable input definitions and user-selected output response metrics are currently limited to those defined in the MSC.Nastran bulk data file (`*.bdf` file) and written to the

MSC.Nastran output file (`*.f06` file). RD automatically generates and manages a number of unique solutions as defined by the analyst and displays the results in several formats to assist the analyst in assessing the design based on the given finite element model.

Solutions that appear to be outliers in the collection or cloud of solutions then need to be examined by the analyst to understand why it is an outlier (*i.e.,* is it something due to a response change or is it an issue related to finite element model limitations or is it a problem that needs to be fixed in the design). The application of the software is geared to having an "engineer in the loop". It is not a tool that will assess the finite element modeling correctness or adequacy, although the ability to run 100 or more analyses using the same finite element discretization provides some confidence in the robustness of the engineering analysis model. It should also be noted that by running these stochastic simulations, a finite element modeling error may be uncovered.

The MSC.Robust Design stochastic simulation tool leverages existing MSC.Nastran finite element models and does not require the generation of new finite element models in order to perform the stochastic simulation. Examination of RD results can provide confidence in the finite element model for the intended application and identify potential design issues related to the design variables (outliers in the design space). By keeping the "engineer in the loop", RD has the potential for increasing design quality by the analyst's understanding of how uncertainties influence the design response using the given analysis model. Establishing the credibility of the given baseline finite element model used in the stochastic simulations still resides with the analyst.

References

1. Sundararahan, C. (editor), *Probabilistic Structural Mechanics Handbook*, Chapman and Hall, New York, 1995.

2. Long, M. W. and Narciso, J. D., *Probabilistic Design Methodology for Composite Aircraft Structures*, Report No. DOT/FAA/AR-99/2, June 1999.

3. Cruse, Thomas A., *Non-Deterministic, Non-Traditional Methods (NDNTM)*, NASA CR-2001-210976, July 2001.

4. Zang, T. A., Hemsch, M. J., Hilburger, M. W., Kenny, S. P., Luckring, J. M., Maghami, P., Padula, S. L., and Stroud, W. J., *Needs and Opportunities for Uncertainty-Based Multidisciplinary Design Methods for Aerospace Vehicles*, NASA TM-2002-211462, July 2002.

5. Anon., *MSC.Nastran Quick Reference Guide, Volumes I and II*, MSC.Software Corporation, Santa Ana, CA, 2005.

6. Anon., *MSC.Robust Design, Version 2004r3 – Installation and User's Guide*, MSC.Software Corporation, Santa Ana, CA, 2004.

7. Allen, Gene, "Robust Design Workshop – Using MSC.Robust Design to Understand Key Drivers and Improve your FE Designs," presented at the *MSC.Software Virtual Product Development Conference*, Huntington Beach, CA, 2004.

8. Curry, D. M., Latchem, J. W, and Whisenhunt, G. B., "Space Shuttle Orbiter Leading Edge Structural Subsystem Development," AIAA Paper No. 83-0483, presented at the AIAA 21st Aerospace Sciences Meeting, Reno, NV, January 10-13, 1983.

9. Curry, Donald M., Johnson, David W., and Kelly, Robert E., "Space Shuttle Orbiter Leading-Edge Flight Performance Compared to Design Goals," in *Shuttle Performance: Lessons Learned*, J. P. Arrington and J. J. Jones (compilers), NASA CP-2283, Part 2, October 1983, pp. 1065-1082.

10. Johnson, D. W., Curry, D. M, and Kelly, R. E., "Space Shuttle Orbiter: Leading Edge Structural Design/Analysis and Material Allowables," AIAA Paper No. 86-0949, presented at the AIAA/ASME/ASCE/AHS/ASC 27th Structures, Structural Dynamics, and Materials Conference, San Antonio, TX, May 19-21, 1986.

11. Curry, Donald M., *Space Shuttle Orbiter Thermal Protection System Design and Flight Experience*, NASA TM-104773, May 1993.

12. Gordon, Michael P., Leading Edge *Structural Subsystem and Reinforced Carbon-Carbon Reference Manual*, Boeing, Report No. KSC-KLO-98-008, October 19, 1998.

13. Gehman, Harold W., Jr. *et al.*, *Columbia Accident Investigation Board Report, Volume 1*, NASA, Washington, DC, August 2003.

14. Smith, T. W. *et al.*, *Leading Edge Structural Subsystem Mechanical Design Allowables for Material with Improved Coating System*, Loral Vought Systems, Report No. 221RP00614 Rev. A, October 12, 1994. Originally released on July 15, 1977.

15. Knight, Norman F., Jr., Moore, David F., and Reeder, James R., *Fracture Mechanics Analyses of Space Shuttle Wing Leading Edge Panel 6L Subjected to Mechanical Loading*, NASA TM-2005-213775, July 2005.

16. Krueger, Ronald, *The Virtual Crack Closure Technique: History, Approach, and Applications*, NASA CR-2002-211628, April 2002. Also available in *Applied Mechanics Reviews*, Vol. 57, No. 2, pp. 109-147, 2004.
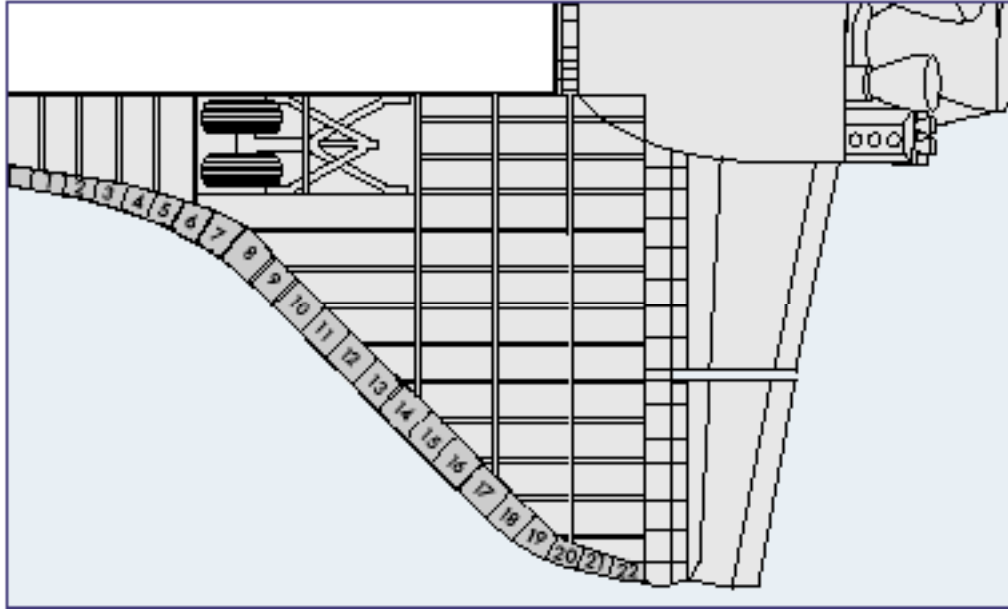
Figure 1. RCC WLE panel numbering on each wing (from CAIB report, Ref. 13, p. 55).
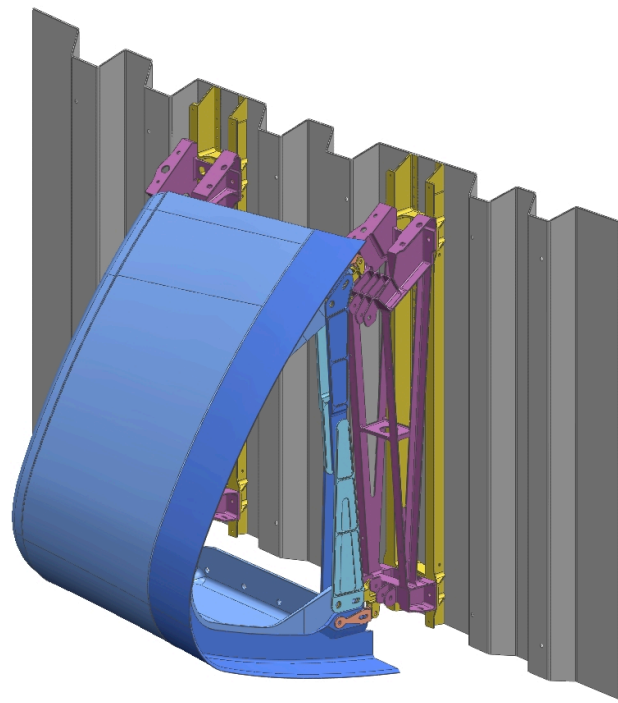
Figure 2. Typical RCC WLE panel assembly (blue parts) and
WLE attachment hardware and front spar (non-blue parts).
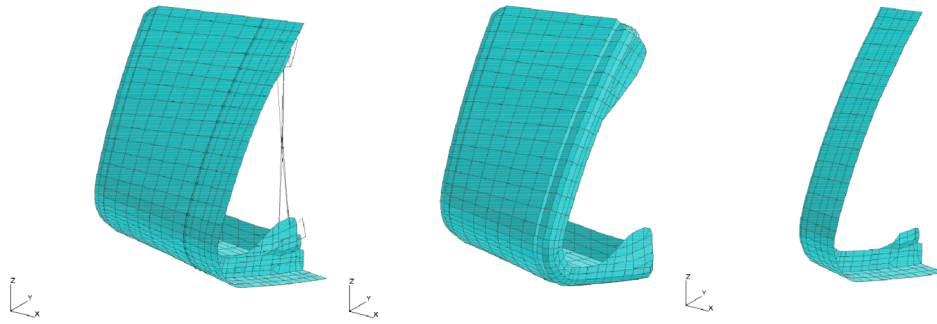
Figure 3. MSC.Nastran shell element model for RCC WLE Panel 4 and T-seal 5
– complete finite element model, panel model only, and T-seal model only (left to right).
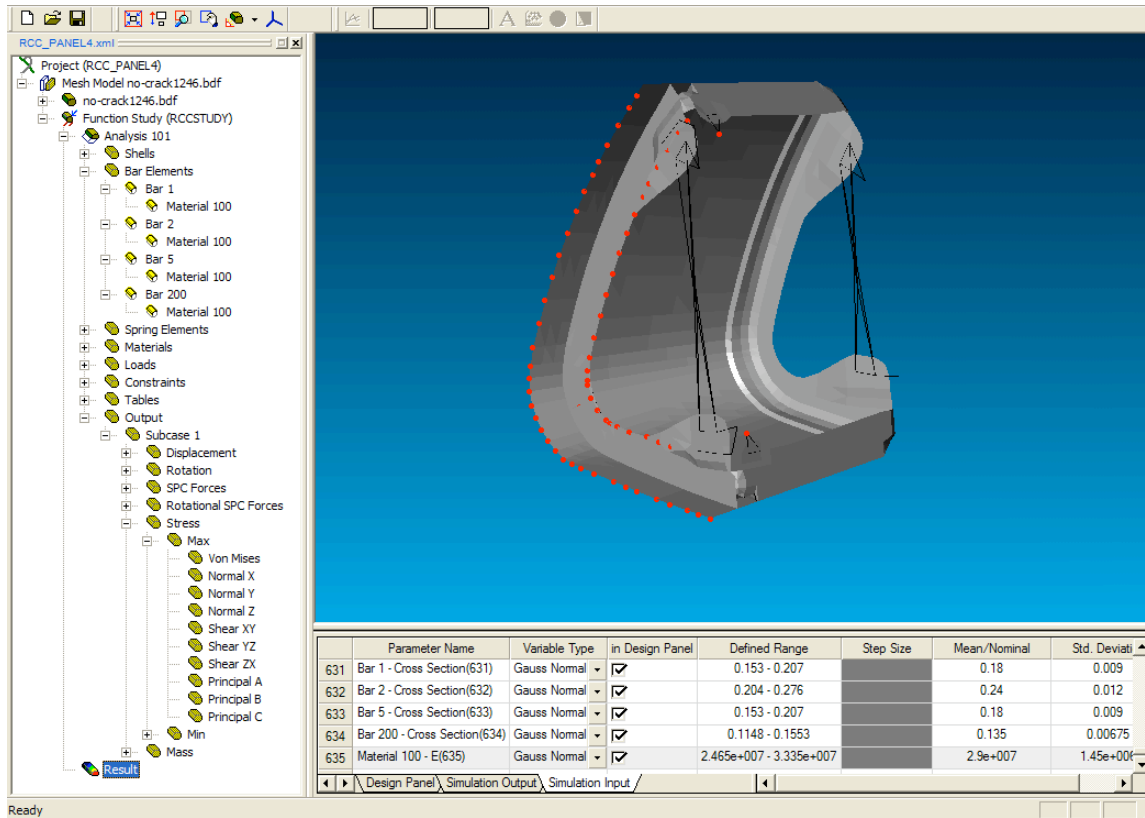
Figure 4. Primary display of MSC.Robust Design after reading
in the MSC.Nastran bulk data file.
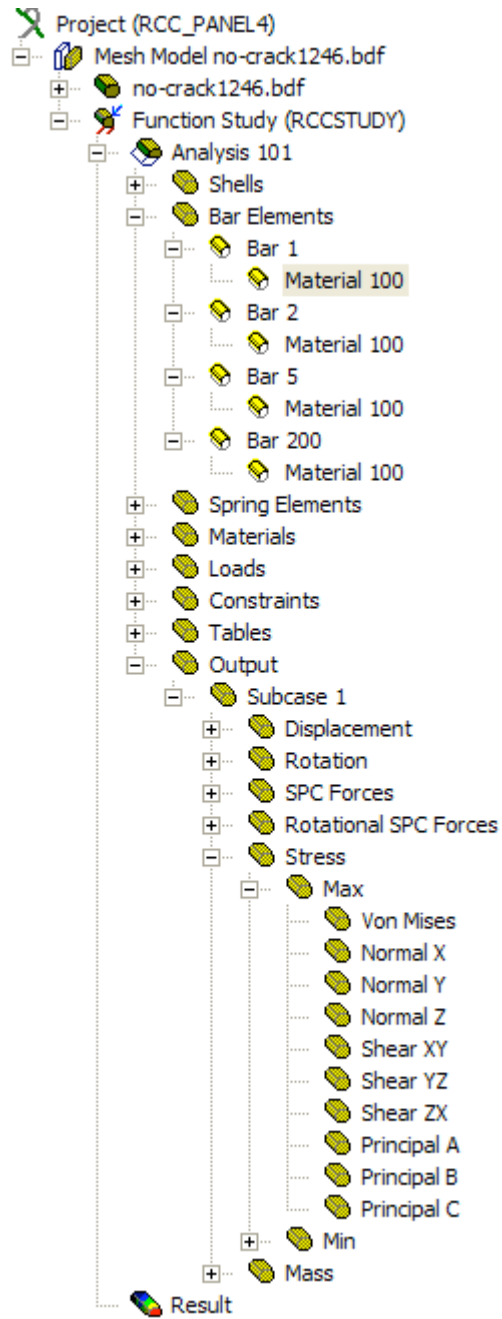
Figure 5. Representative "Project Tree" formed by MSC.Robust Design
upon reading the MSC.Nastran bulk data file.
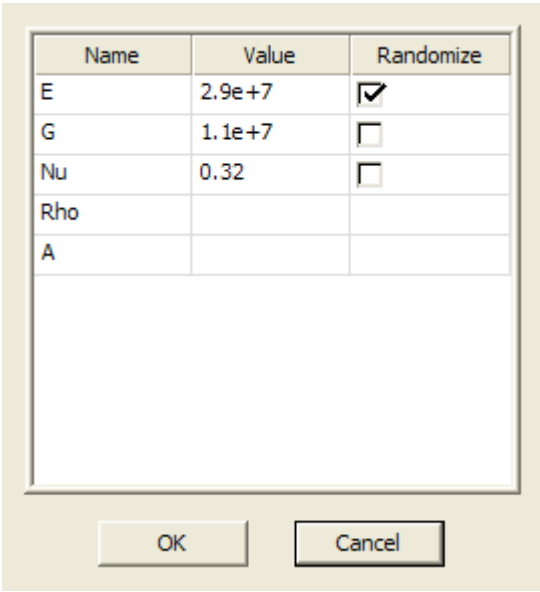
(a) Selecting a single material parameter to be randomized.
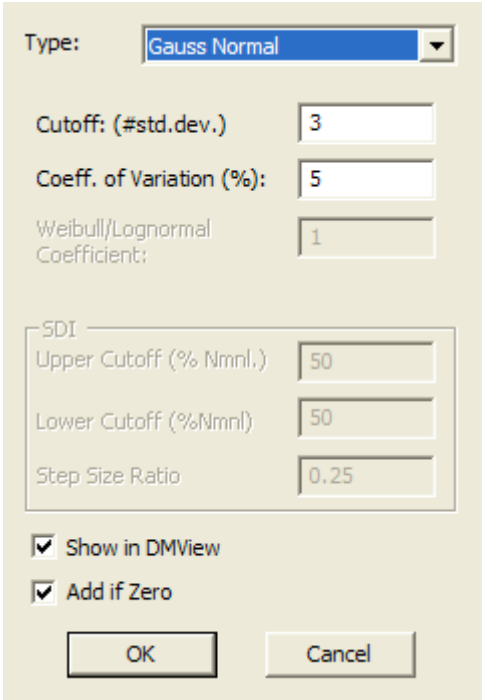
(b) Setting the distribution type, cutoff, and coefficient of variation.

Figure 6. Randomizing input design variables for the stochastic
simulation using MSC.Robust Design.
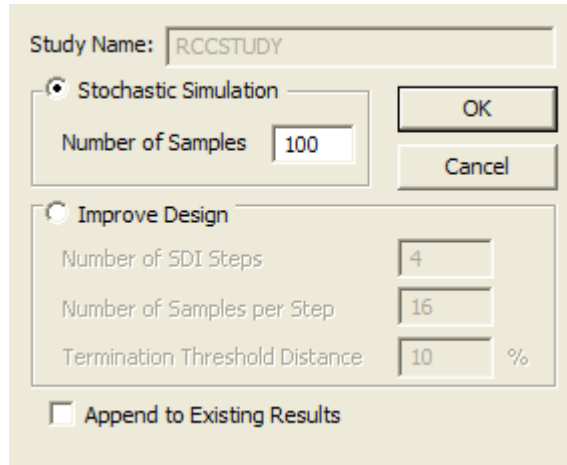
Figure 7. Setting the number of stochastic simulations (samples) for MSC.Robust Design.
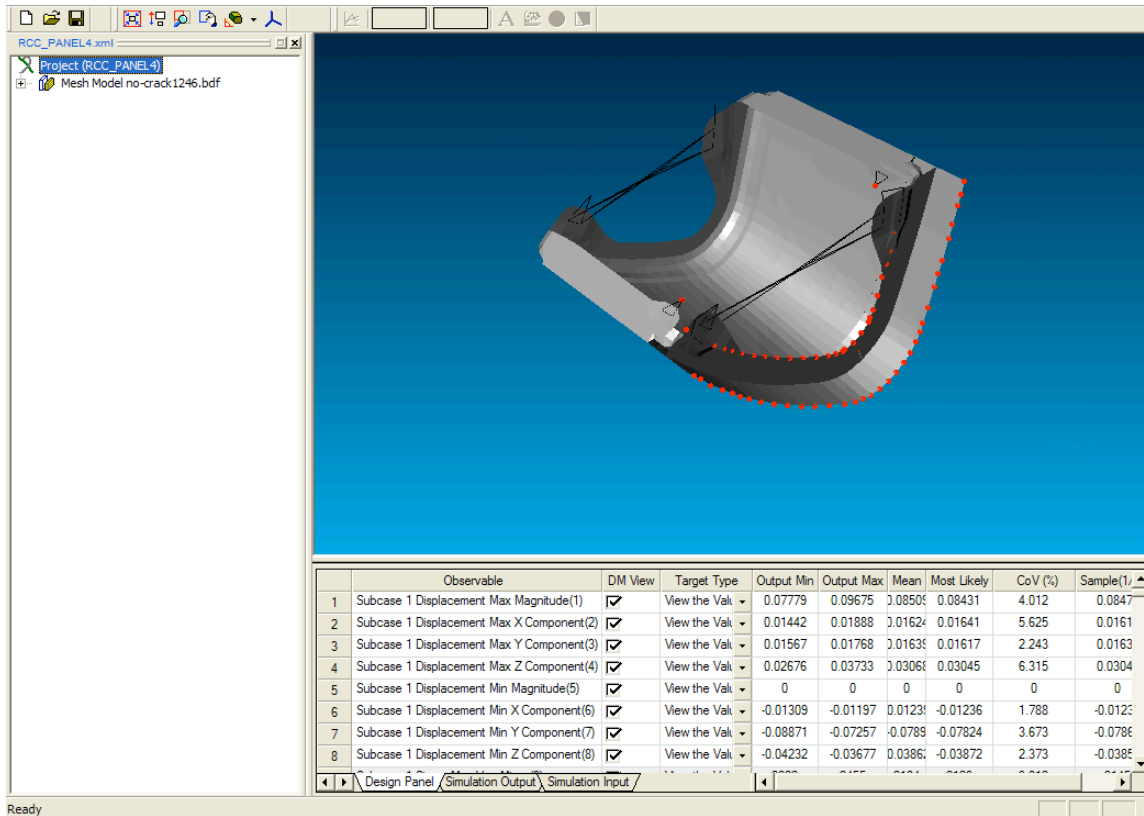
Figure 8. MSC.Robust Design display of the Design Panel tab
showing typical simulation results.
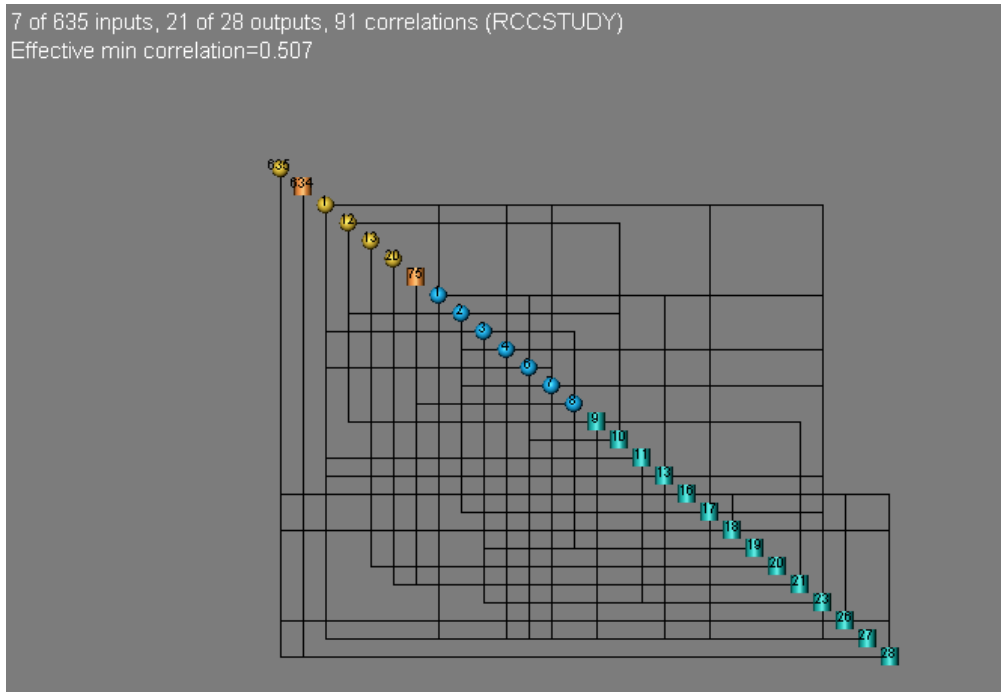
(a) Decision map without labels.
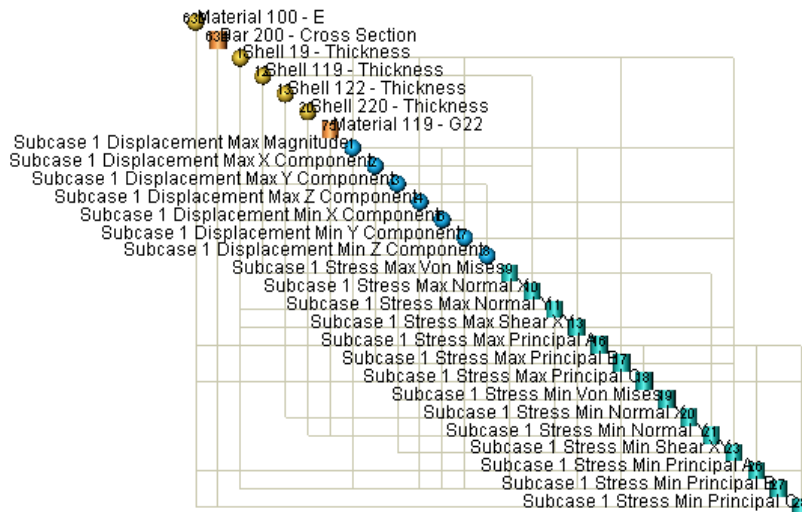
(b) Decision map with labels.

Figure 9. Representative decision map from MSC.Robust Design for specified correlation factor r_0 of 0.50.
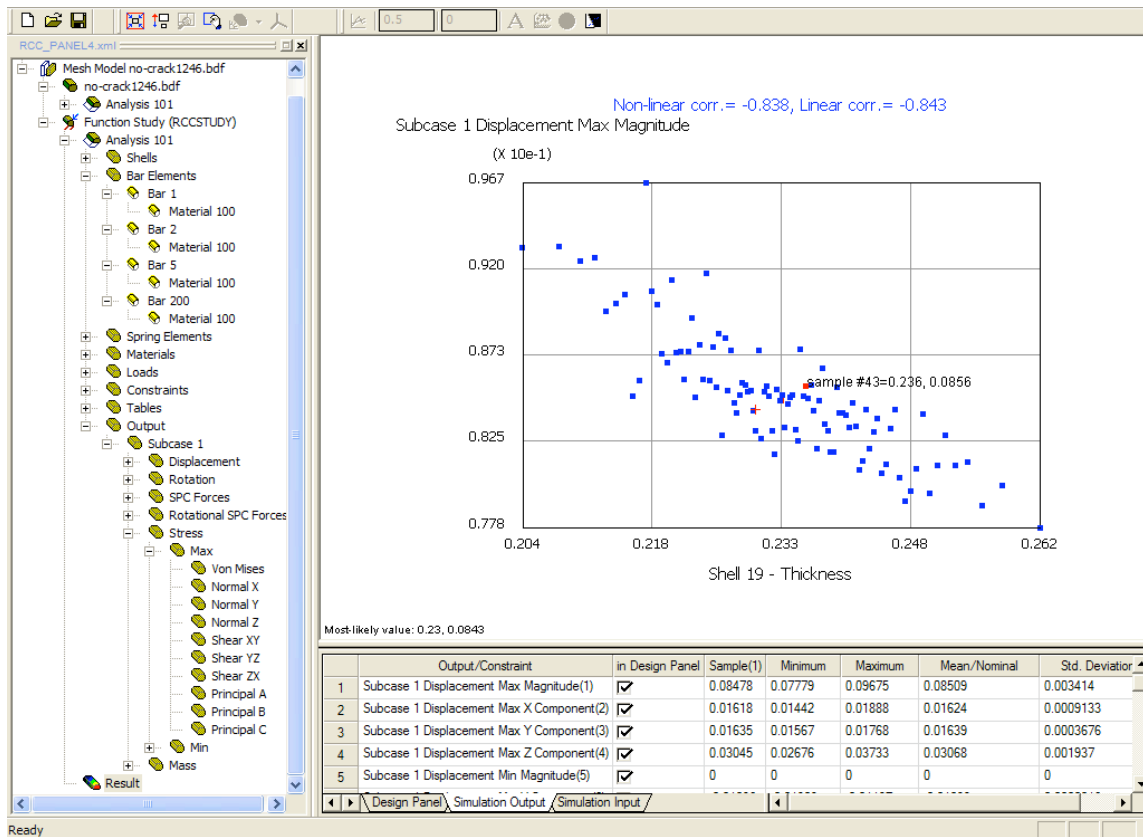
Figure 10. Representative scatter plot or cloud from MSC.Robust Design.
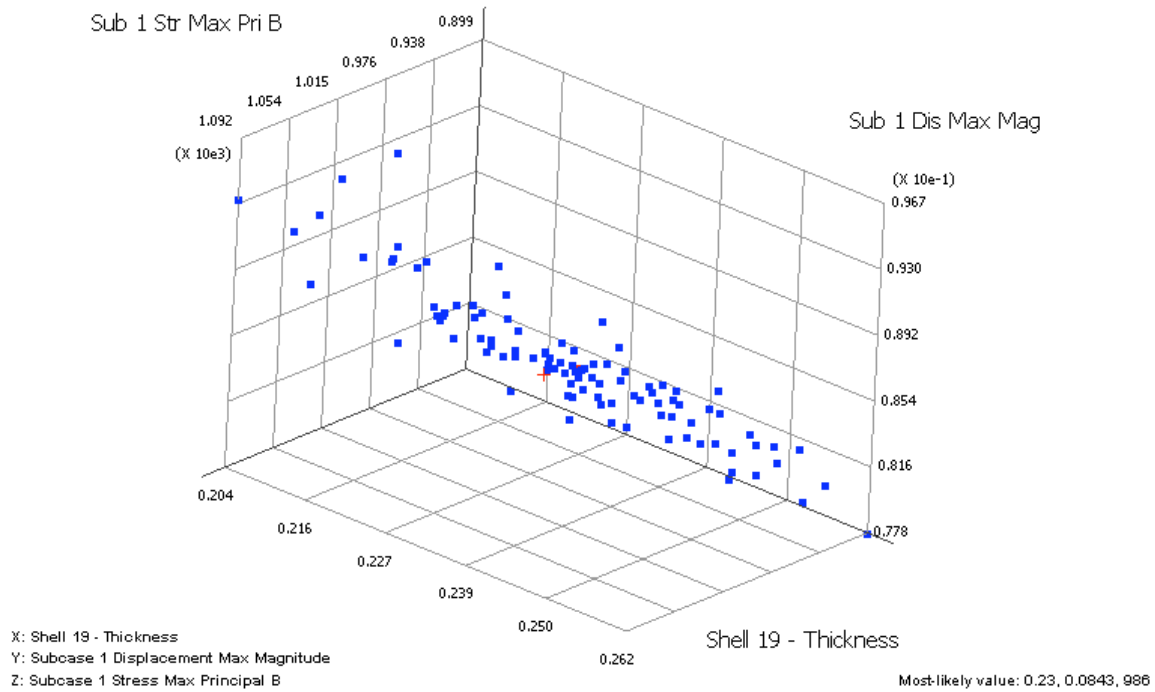
Figure 11. Representative three-dimension ant-hill plot – shell thick, maximum displacement magnitude, maximum principal stress B.
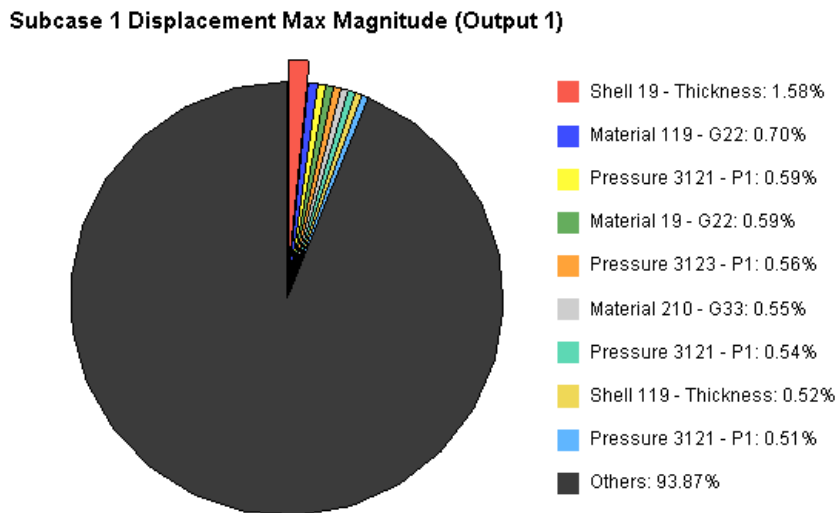
Figure 12. Typical pie-chart distribution for specific output variable in terms of the input variables.
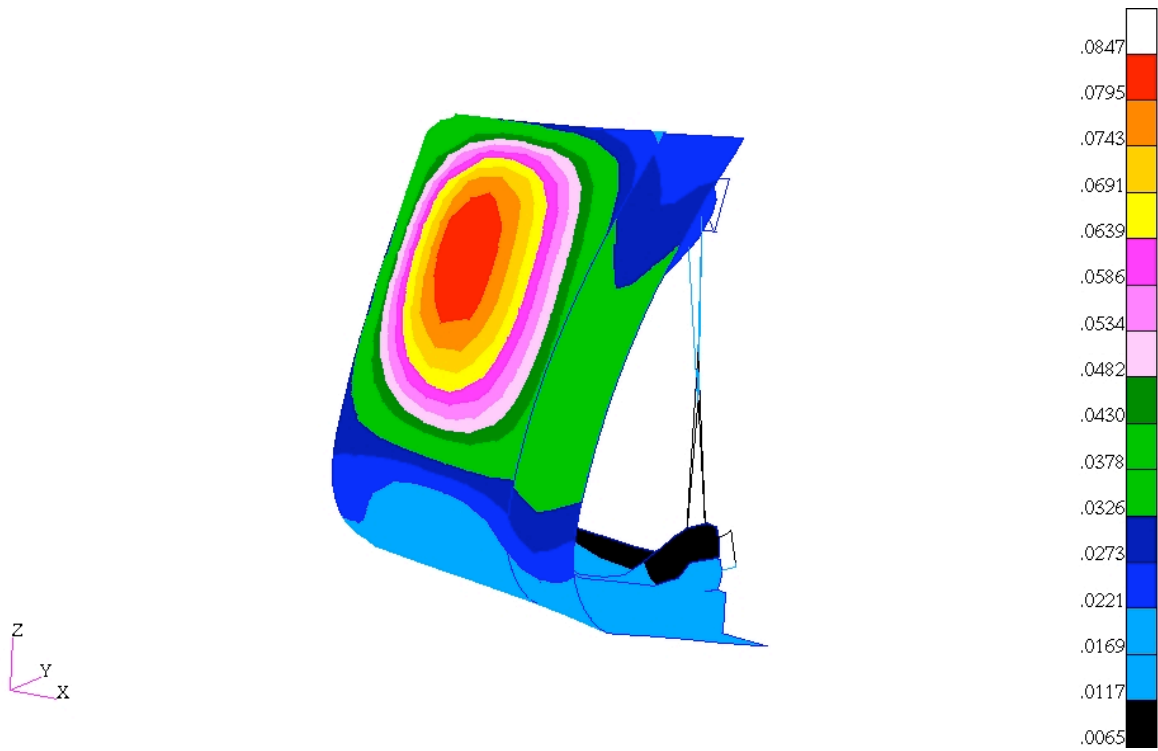
Figure 13. Contour plot of the resultant displacement magnitudes for the RCC WLE Panel 4 assembly subjected to a re-entry aerodynamic load case, in inches.
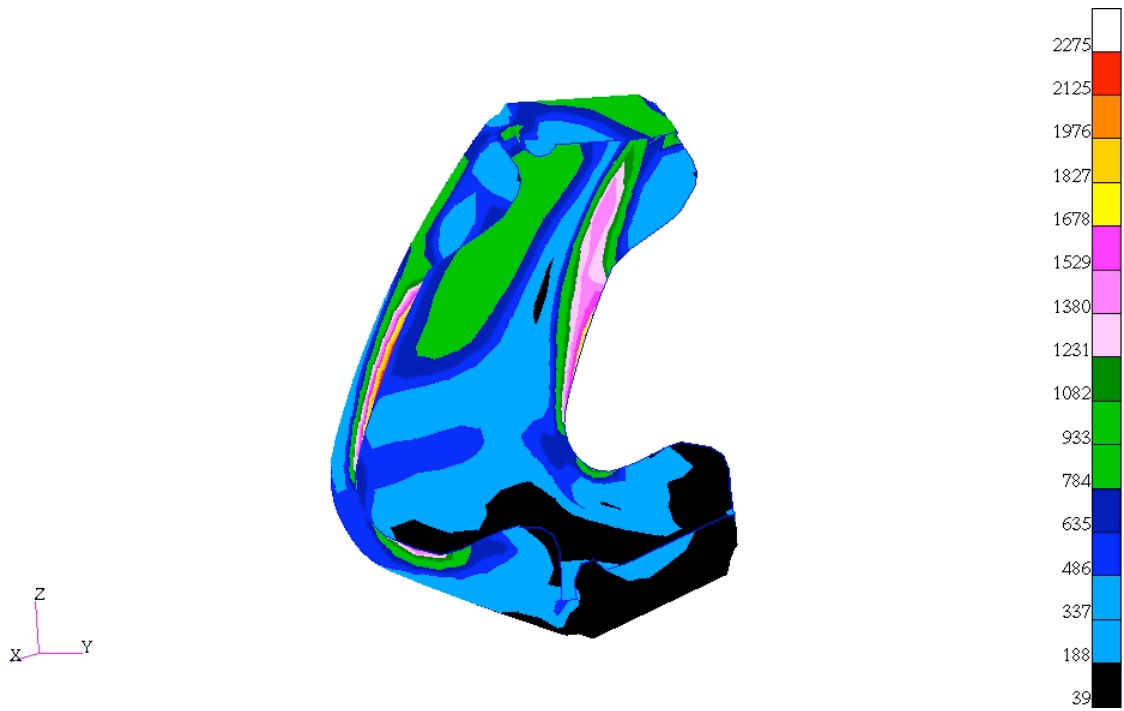
(a) Stress distribution on the panel, in psi.
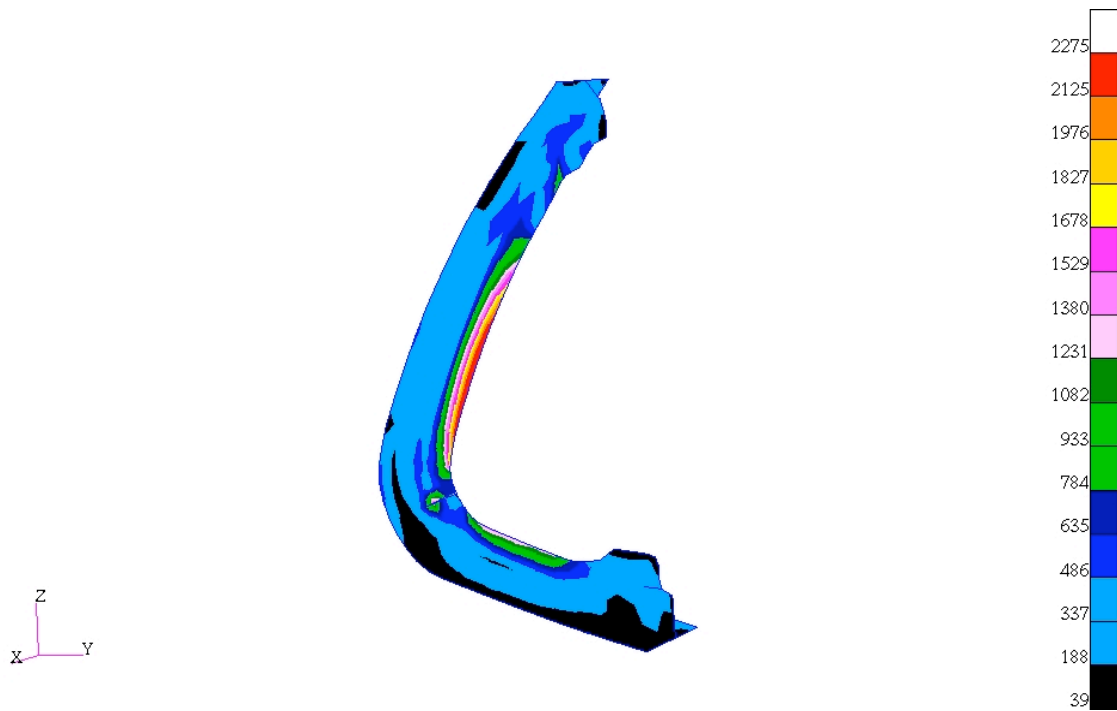
(b) Stress distribution of the T-seal, in psi.

Figure 14. Contour plots of the von Mises stress distributions on the outer surface (Z1) of
RCC WLE Panel 4 assembly subjected to a re-entry aerodynamic load case.
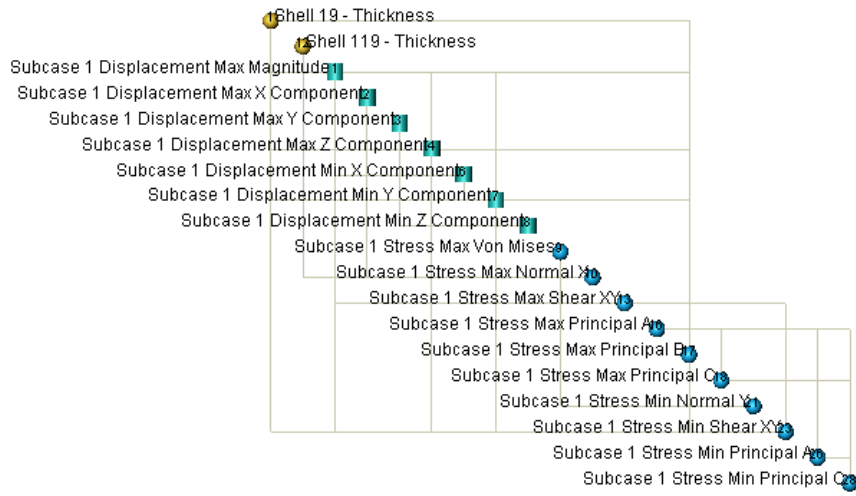
(a) Increasing the specified correlation factor to 0.75.
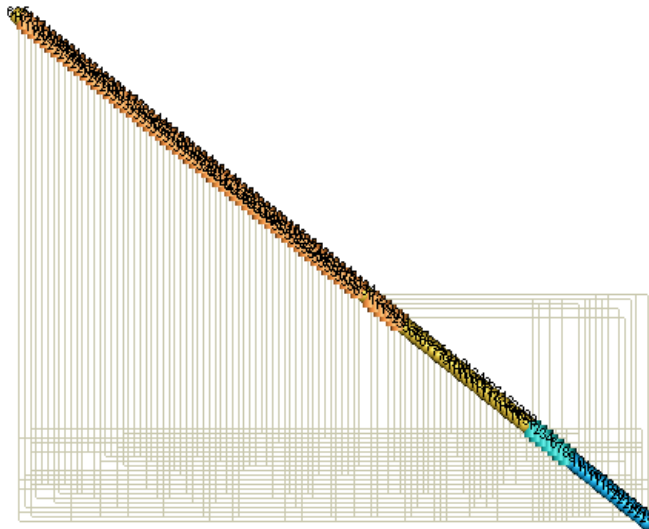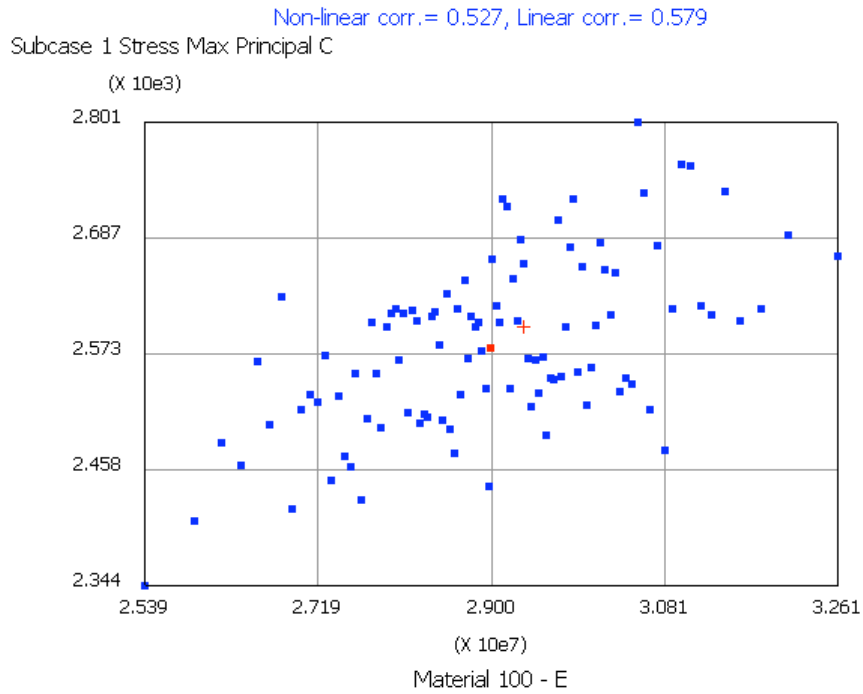
(b) Decreasing the specified correlation factor to 0.25.

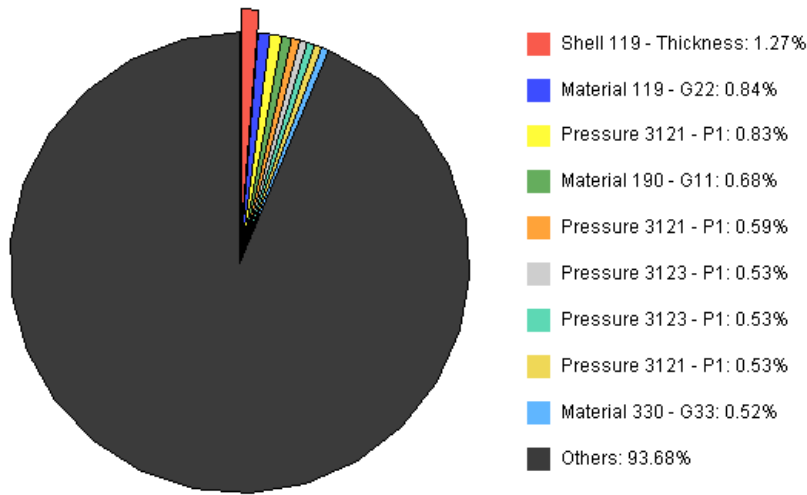Figure 15. Effect of specified correlation factor r_0 on decision map.
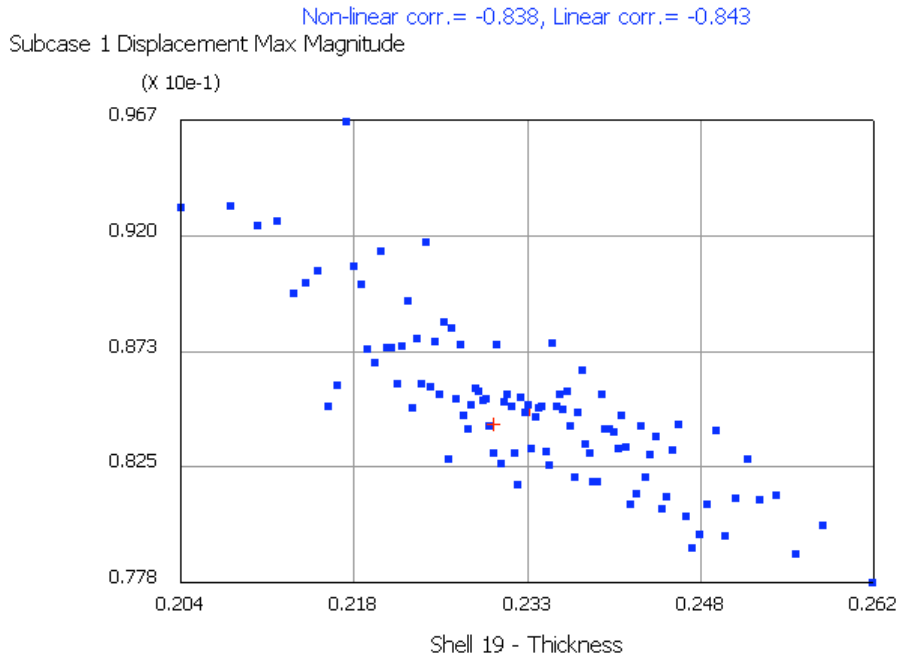
Figure 16. Representative scatter plot of displacement results.

Figure 17. Representative scatter plot of stress results.

Subcase 1 Stress Max Von Mises (Output 9)

- Shell 119 - Thickness: 1.27%
- Material 119 - G22: 0.84%
- Pressure 3121 - P1: 0.83%
- Material 190 - G11: 0.68%
- Pressure 3121 - P1: 0.59%
- Pressure 3123 - P1: 0.53%
- Pressure 3123 - P1: 0.53%
- Pressure 3121 - P1: 0.53%
- Material 330 - G33: 0.52%
- Others: 93.68%

(a) Pie-chart distribution for the maximum von Mises stress.

Subcase 1 Displacement Max Magnitude (Output 1)

- Shell 19 - Thickness: 1.58%
- Material 119 - G22: 0.70%
- Pressure 3121 - P1: 0.59%
- Material 19 - G22: 0.59%
- Pressure 3123 - P1: 0.56%
- Material 210 - G33: 0.55%
- Pressure 3121 - P1: 0.54%
- Shell 119 - Thickness: 0.52%
- Pressure 3121 - P1: 0.51%
- Others: 93.87%
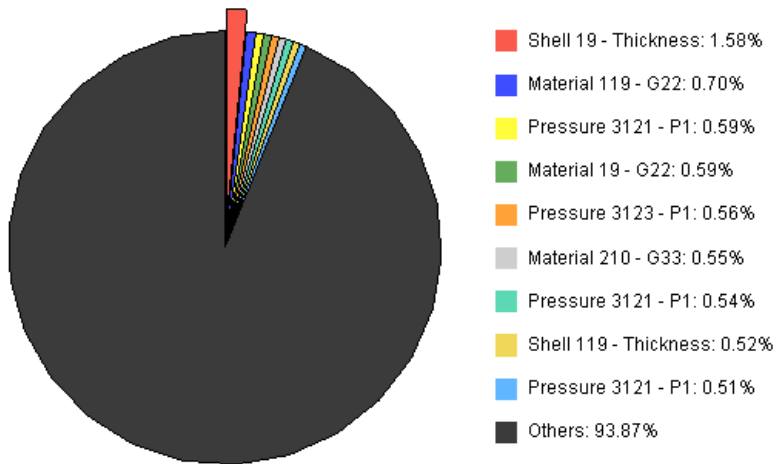
(b) Pie-chart distribution for the maximum resultant displacement.

Figure 18. Representative pie-chart distributions for different response output variables.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. REPORT DATE *(DD-MM-YYYY)*	2. REPORT TYPE	3. DATES COVERED *(From - To)*
01- 08 - 2006	Technical Memorandum	

4. TITLE AND SUBTITLE	5a. CONTRACT NUMBER
Stochastic Simulation Tool for Aerospace Structural Analysis	
	5b. GRANT NUMBER
	5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S)	5d. PROJECT NUMBER
Knight, Norman F., Jr.; and Moore, David F.	
	5e. TASK NUMBER
	5f. WORK UNIT NUMBER
	377816.06.03.03.03

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)	8. PERFORMING ORGANIZATION REPORT NUMBER
NASA Langley Research Center Hampton, VA 23681-2199	L-19277

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)	10. SPONSOR/MONITOR'S ACRONYM(S)
National Aeronautics and Space Administration Washington, DC 20546-0001	NASA
	11. SPONSOR/MONITOR'S REPORT NUMBER(S)
	NASA/TM-2006-214500

12. DISTRIBUTION/AVAILABILITY STATEMENT

Unclassified - Unlimited
Subject Category 39
Availability: NASA CASI (301) 621-0390

13. SUPPLEMENTARY NOTES

Knight: General Dynamics - Advanced Information Systems; Moore, Langley Research Center
An electronic version can be found at http://ntrs.nasa.gov

14. ABSTRACT

Stochastic simulation refers to incorporating the effects of design tolerances and uncertainties into the design analysis model and then determining their influence on the design. A high-level evaluation of one such stochastic simulation tool, the MSC.Robust Design tool by MSC.Software Corporation, has been conducted. This stochastic simulation tool provides structural analysts with a tool to interrogate their structural design based on their mathematical description of the design problem using finite element analysis methods. This tool leverages the analyst's prior investment in finite element model development of a particular design. The original finite element model is treated as the baseline structural analysis model for the stochastic simulations that are to be performed. A Monte Carlo approach is used by MSC.Robust Design to determine the effects of scatter in design input variables on response output parameters. The tool was not designed to provide a probabilistic assessment, but to assist engineers in understanding cause and effect. It is driven by a graphical-user interface and retains the "engineer-in-the-loop" strategy for design evaluation and improvement. The application problem for the evaluation is chosen to be a two-dimensional shell finite element model of a Space Shuttle wing leading-edge panel under re-entry aerodynamic loading. MSC.Robust Design is easy to use and adds value to the analysis effort by rapidly being able to identify design input variables whose variability cause the most influence in response output parameters.

15. SUBJECT TERMS

Design; Finite Elements; Stochastic Simulation

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT	b. ABSTRACT	c. THIS PAGE			STI Help Desk (email: help@sti.nasa.gov)
U	U	U	UU	39	19b. TELEPHONE NUMBER *(Include area code)* (301) 621-0390

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std. Z39.18