NASA/TM-20240000075

Forced Response Aeromechanics Analysis in MATLAB®-Based Environment Code With Application to Distortion-Tolerant Fan R24 Blade Geometry

Austin J. Schemmel
Glenn Research Center, Cleveland, Ohio

February 2024

NASA STI Program Report Series

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA scientific and technical information (STI) program plays a key part in helping NASA maintain this important role.

The NASA STI program operates under the auspices of the Agency Chief Information Officer. It collects, organizes, provides for archiving, and disseminates NASA's STI. The NASA STI program provides access to the NTRS Registered and its public interface, the NASA Technical Reports Server, thus providing one of the largest collections of aeronautical and space science STI in the world. Results are published in both non-NASA channels and 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 has 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.

- 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 also include organizing and publishing research results, distributing specialized research announcements and feeds, providing information desk and personal search support, and enabling data exchange services.

For more information about the NASA STI program, see the following:

- Access the NASA STI program home page at http://www.sti.nasa.gov

NASA/TM-20240000075

Forced Response Aeromechanics Analysis in MATLAB®-Based Environment Code With Application to Distortion-Tolerant Fan R24 Blade Geometry

Austin J. Schemmel
Glenn Research Center, Cleveland, Ohio

National Aeronautics and
Space Administration

Glenn Research Center
Cleveland, Ohio 44135

February 2024

Acknowledgments

This work was completed with the support of the Advanced Air Transport Technology Project and the Transformational Tools and Technologies Project. The author greatly appreciates the support of both projects, as well as that of Dr. Milind Bakhle for his technical expertise and advice during code development.

This work was sponsored by the Advanced Air Vehicles Program at the NASA Glenn Research Center.

This work was sponsored by the Transformative Aeronautics Concepts Program.

Level of Review: This material has been technically reviewed by technical management.

This report is available in electronic form at https://www.sti.nasa.gov/ and https://ntrs.nasa.gov/

NASA STI Program/Mail Stop 050
NASA Langley Research Center
Hampton, VA 23681-2199

Forced Response Aeromechanics Analysis in MATLAB®-Based Environment Code With Application to Distortion-Tolerant Fan R24 Blade Geometry

Austin J. Schemmel
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

Summary

Researchers at the NASA Glenn Research Center have developed a research-level code for forced response analysis for turbomachinery that provides a streamlined framework for aeromechanics analysis, as well as for generating Goodman diagrams. The Forced Response Aeromechanics Analysis in a MATLAB-Based Environment (FRAAME, Version 1) code is designed to accept blade surface unsteady pressure time histories generated by three-dimensional (3D) Unsteady Reynolds-Averaged Navier-Stokes (URANS) code TURBO and modal displacements, modal stresses, and static stresses generated via expanded ANSYS® (Ansys, Inc.) cyclic symmetry analysis. The code's looping structure allows for large-scale analyses, including many blade surface pressure files, modal displacements, and modal stress files for full annulus forced response analysis, including a modal summation method for multiple engine orders and modes. This code is applied to the R24 fan geometry for the Boundary Layer Ingesting Inlet/Distortion-Tolerant Fan (BLI^{2}DTF) project, a propulsion system being developed to increase the fuel efficiency of future aircraft, to serve as validation of experimental data as well as external simulation results.

This tool's functional, modular form is intended to allow users to make modifications easily. Those modifications can include requests for information for any step in the analysis process, as well as adding various functions to compute additional information of interest. It is also written, given sufficient input information, as a general blade geometry forced response solver that is not necessarily specific to the R24 geometry but is used here for verification of code functionality and accuracy. Although forced response analysis tools are also available in commercially available code packages, the post-processing FRAAME code offers reliable, fast, and scalable forced response analysis and Goodman diagram generation for large cases utilizing, specifically, TURBO and ANSYS® results information in an effort to streamline forced response and high cycle fatigue analyses.

Nomenclature

3D	three-dimensional
asf	ANSYS® scaling factor
ANSYS®	commercial FEA solver
BLI	boundary layer ingestion
BLI^{2}DTF	Boundary Layer Ingesting Inlet/Distortion-Tolerant Fan
Campbell diagram	plot that represents a system's response spectrum as a function of its oscillation regime
CFD	computational fluid dynamics

Delaunay triangulation	interpolation algorithm for mapping modal displacements to unsteady blade surface pressure grid
DTF	distortion-tolerant fan
FEA	finite element analysis
FFT	fast Fourier transform
FRAAME	Forced Response Aeromechanics Analysis in a MATLAB-Based Environment
Goodman diagram	graphical representation of the risk of high cycle fatigue failure of a structural component
URANS	Unsteady Reynolds-Averaged Navier-Stokes

Introduction

The complex nature of fluid-structure interaction demands taking particular care in turbomachinery component design where components such as engine fans can be exposed to large dynamic loading phenomena. Dynamic effects on the structures include vibratory loading due to rotational loads, as well as aerodynamic loads during operation. Aeroelastic phenomena such as forced response and flutter, which are produced by turbomachine operating conditions, can lead to high cycle fatigue and sudden catastrophic failure. Development of a tool to predict forced response and high cycle fatigue is required so that areas of expected resonance or modal contributions to a structure's overall vibration pattern can be analyzed in an effort to minimize the chances of encountering aeroelastic effects during operation that can lead to structural failure. Various commercial code packages possess the capability of performing forced response analysis internally depending on compatibility and availability of computational fluid dynamics (CFD) and finite element analysis (FEA) solvers. Researchers at NASA Glenn Research Center utilize TURBO, a noncommercial CFD code developed in house (Refs. 1 and 2) for aeromechanics analysis, in conjunction with commercial FEA solvers ANSYS® (Ansys, Inc.) and MSC Nastran (Hexagon AB). Recently, the team at NASA Glenn used the MATLAB® (MathWorks, Inc.) programming language to create an external post-processing tool, Forced Response Aeromechanics Analysis in a MATLAB-Based Environment (FRAAME, Version 1) code, which provides a streamlined framework capable of coupling TURBO/ANSYS® output formats in a batch processing utility. The primary advantage of this framework is repeatability for potential design changes, CFD/FEA run condition changes, etc. Output from each solver can be post processed for forced response solutions and Goodman diagram generation for batch jobs containing unsteady aerodynamics results for multiple blade passages and structural dynamics solutions for multiple modes/engine orders with minimal user input. Similarly, specific conditions can be isolated and analyzed individually if a special case is of interest including, for example, a mode/engine order Campbell diagram crossing, which may correspond to a large dynamic response during operation. Another benefit of a separate post-processing environment is immediate generation of new solutions in the event potential changes arise in CFD/FEA analyses. For example, if changing CFD flow conditions or inputs require rerunning cases, then forced response analysis must be completely restarted. The commercial solution currently in use can perform forced response analysis but only on a case-by-case basis including a single mode/engine order. In contrast, FRAAME offers a one-step environment in which successive CFD runs can be deposited into the working directory and a batch job of updated forced response analyses for multiple modes/engine orders can be completed using the existing modal analysis results.

This report documents the code development considerations for general geometries, as well as the code's application to a real-world case that was tested previously in NASA Glenn's 8- by 6-Foot Supersonic Wind Tunnel (Ref. 3). This report outlines FRAAME development efforts, as well as the code's application to specific cases pertaining to analysis performed for the R24 Distortion-Tolerant Fan (DTF) for purposes of validating forced response analysis and Goodman diagram generation.

Description of Forced Response Analysis Solvers

Forced response analysis and Goodman diagram generation effectively require fundamental inputs of unsteady aerodynamics information, modal displacements, and modal and static stresses. Unsteady aerodynamic information used in this FRAAME code is generated by TURBO, a time-domain, three-dimensional (3D) Unsteady Reynolds-Averaged Navier-Stokes (URANS) solver that is used to simulate fan operation. TURBO simulations result in unsteady blade surface pressure data as a time history that is used in this code as input. Structural dynamics information is generated by the ANSYS® commercial finite element structural analysis solver. ANSYS® simulations result in modal displacements and modal and static stresses, which are used in this code as input. A neglected consideration for this effort that will be explored in further work includes the effects of blade mistuning, in which small differences in blade-to-blade geometry may influence the flow field and blade natural frequencies. Therefore, this code analyzes uniform blade geometry for all blades around the annulus. This code is written to analyze all blades around the annulus given sufficient input information. The application of this code for this report considers pressure information for a single blade in the model, as well as a single blade from an expanded model using cyclic symmetry structural dynamics analysis. The code imports a file generated by TURBO that contains CFD model information and the blade surface unsteady pressure time history. The code also imports various files generated by ANSYS®, which include full model information, modal displacements, modal stresses, static stresses, and frequency information, depending on which modes/nodal diameters the user requests to be analyzed. Once information is imported and extracted, a generalized forced response parameter is calculated for each mode at each engine order for each blade considered. This information is then applied to the modal stresses to generate a Goodman diagram for each mode at each engine order for each blade considered. Ultimately, a modal summation can be performed to generate a Goodman diagram that includes all modal contributions for each engine order. This represents each vibration mode's contribution to the high cycle fatigue of the structure, depending on the operating conditions prescribed as inputs to the main script including, for example, part speed conditions. The application of this code to the R24 fan geometry provides repeatable, accurate forced response analysis for various part speed conditions with resonant and nonresonant responses.

Code Development: Case-Specific Input

In addition to the input files required to perform a forced response analysis, some specific inputs are also required when using the FRAAME code. In particular, four manual inputs must be set for each simulation. Inputs from CFD analysis, FEA analysis, and manual inputs are processed by the code's main loop to produce the outputs shown in Figure 1.

The first of the four required manual inputs is a CFD input that considers flow field periodicity. TURBO simulations typically involve many fan blade revolutions in order to ensure confidence that the flow field is periodic and the solution is numerically converged. A revolution selection input can capture individual or multiple revolutions in the unsteady pressure time history. In this analysis procedure, the final revolution is selected because it contains a periodic solution from the previous revolution.

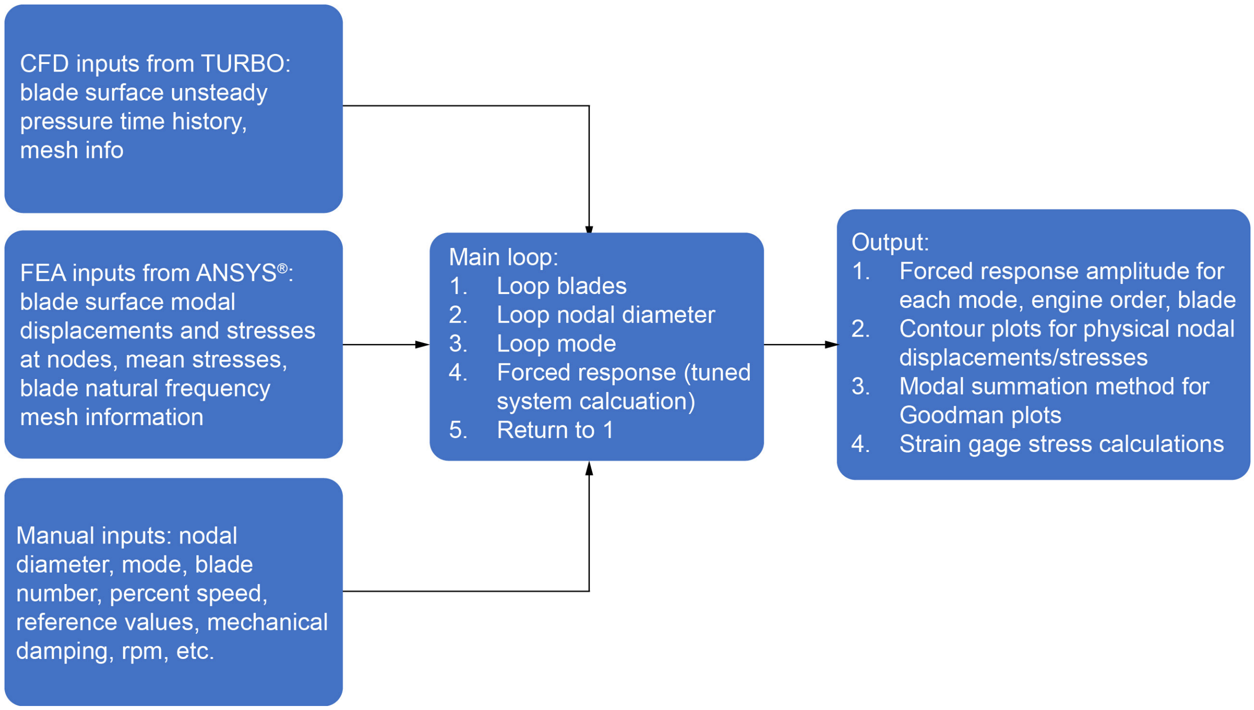

Figure 1.—FRAAME code logic flowchart.

The second input is an FEA input that considers the ANSYS® solution model. Cyclic symmetry analysis includes either an expanded or unexpanded model, in which the expanded model is more representative of the full fan configuration, and the unexpanded model is more representative of a single blade sector. This specification involves a scaling factor that must be applied to all modal displacements and stresses during forced response analysis. The expanded model will include a scaling factor as a function of selected nodal diameter; the unexpanded model will not. A final consideration in this logic includes the nodal diameter that is selected. For example, a fan containing 12 blades has the same scaling factor for nodal diameters 0 and 6. All other nodal diameters have the same scaling factor but different than nodal diameters 0 and 6. This input currently invokes several similar functions in the main script, which are called individually, depending on which case is used. For the purposes of the analysis presented in this report, modes 1 through 3 with nodal diameters 1 and 4 are considered for the expanded cyclic symmetry model, so only one function is utilized.

The third input is a true-false flag for specifying the blade natural frequency for the mode of interest. A specification of blade natural frequency is usually not required, but a certain consideration when calculating forced response is a worst-case approach where the forcing frequency due to the aerodynamic loading and the natural vibration frequency of the blade are exactly the same. This produces the largest response on the blade as a function of frequencies and mechanical/aerodynamic damping. This corresponds to the largest response considered for specific conditions, namely, rpm/natural frequency/engine order/mode crossings on a Campbell diagram. If the flag is true, a partner input includes a specific natural frequency that can be used to modify natural frequency if experiment differs from simulation. If the flag is false, the partner input can be left as any arbitrary value, and the code will calculate the forcing frequency as a function of percent speed and selected nodal diameter. For the purpose of this code, a statement is embedded that checks frequencies and will set them equal if they are within a 10 percent margin in order to capture the largest possible response; this provides a conservative estimate on calculated stresses.

The fourth input considered allows the user to specify whether strain gage information is included as FEA node numbers or nodal coordinates. A separate function determines how to identify candidate FEA node numbers for given gage locations or calculates gage stresses given FEA node numbers. For the purpose of this analysis, gage locations and FEA node numbers are known and both are considered in an effort to compare with experimental and simulation results in Reference 4.

The remaining inputs required for code functionality are case specific. Nodal diameter, mode, and blade number are specified and act as the main driver of the forced response analysis as inputs to the main logic loop. Additional case-specific manual inputs include blade percent speed, timesteps per revolution, binary file header length, number of blades in the model, structural damping, revolutions per minute, strain gage FEA node number and/or nodal coordinates, and frequency margin. For one of the R24 conditions presented in this report, a resonant response is of interest as it was observed in experiment. In order to capture the full effect of the resonant response, that is, when the forcing frequency is equal to the blade natural frequency for a specific mode, a frequency margin is established as a check in the event a forcing frequency is close to the natural frequency. Other manual inputs that are required for TURBO simulations, including blade length scale for dimensionalization of the CFD mesh, reference pressure, and air specific heat, are available in TURBO output files with naming convention 'TURBO.#.out', but they are currently required for this code as manual input. FRAAME code output includes mode-specific forced response calculations that can be applied to modal information to produce physical displacements/stresses for the model. The steps that follow include plotting contours, Goodman diagram generation via modal summation method, and stress calculations at prescribed strain gage locations.

Code Development: CFD Information

TURBO, a time-domain, 3D URANS solver, is used to model fan flow. The R24 fan blade geometry is designed specifically to operate in an environment in which large inlet distortions interact with the fan. The inflow boundary conditions used in this analysis are due to boundary layer ingestion (BLI) inflow as the design considers an airframe embedded engine. The TURBO simulation with BLI distortion results in unsteady blade surface pressure time history data that can be exported with the naming format 'prhist.#.hs' for any blade in the model. The default format of the written files includes an unformatted binary Fortran syntax, which for the initial purposes of this work required a format conversion to formatted stream, as well as variable changes from integer to double precision in order for the MATLAB® programming environment to extract the data more easily. Due to the format of the Fortran binary files, careful consideration was necessary while importing each pressure history file, including consideration of header information in which Fortran-compatible file readers natively take account. Information of interest that is written to the pressure history files includes grid index information, number of timesteps completed for each TURBO simulation, nondimensional grid surface coordinates, nondimensional grid cell areas, and unsteady pressures distributed onto grid cell faces for all timesteps and revolutions requested for each TURBO simulation. The extracted information that is required for forced response analysis can be used to calculate a pressure force time history on the blade surface as a preliminary step if required. This routine serves as an automated approach to extracting relevant CFD information for analysis and can be updated depending on available pressure history files.

Code Development FEA Information

The FRAAME tool uses ANSYS® modal analysis results as input, namely modal displacements and both modal and static stresses. ANSYS® can generate a text file including the full FEA node numbering and corresponding 3D Cartesian nodal coordinates; a function within the FRAAME code imports and extracts this information in one step. This information is used in the main loop to filter coordinates from only information on the blade surface from the modal analysis files. Currently in this code, FEA results of interest are imported as readable text files as opposed to CFD results, which are written in a binary format. Naturally, each text file will include extraneous information that is not considered useful for analysis, such as text indicating job name, headers, etc. Given that the information of interest is mostly numerical, a specified string of ASCII characters including text is defined and used to filter out any extraneous text in the file to make organizing data a simpler process. After this extraneous text is removed, a fixed-width field delimiter consistent with the ANSYS® output structure converts the text strings to double-precision usable data. This process allows for a generalized, automated approach to reading various files containing modal information for different modes and nodal diameters that are used in this analysis. Each step in importing and organizing FEA data uses this approach assuming consistent file nomenclature. Once the aerodynamics information is extracted, a function extracts modal displacements and computes forced response. This function searches for files with a set nomenclature and specific string information including blade geometry for any mode and nodal diameter requested. A list of strings included as manual input representing keywords determine the modal information and blade natural frequencies in each file. Once nodes and corresponding modal displacements are extracted, coordinates are filtered from the full model such that the blade surface nodes, coordinates, and modal displacements are saved for further aerostructure computations. Similar logic is applied in another function after forced response calculations in the main loop that handle modal stresses. The naming convention for modal displacements and stresses are intentionally made similar so that file-reading logic can be applied to both functions with minor changes. A special consideration is made for the R24 case in which variation in materials at the blade and disk potentially influence results. In other words, there is a distinct possibility that the connection between the blade and disk may contain shared node numbers in which the stresses associated with the shared node number are averaged. Given that most analysis results for this case indicate larger deformations and stress concentrations through the blade in the spanwise direction, averaging nodes near the hub most likely will not influence the solution by a considerable amount. The information at the shared nodes of the blade and disk are averaged for the computation to continue. Ultimately, this modal stress function produces scaled complex stresses at each node on the blade surface, as well as Von Mises stresses that will contribute to generated Goodman diagrams. Another function using similar logic, the modal displacement and stress functions, is used to calculate static Von Mises stresses, which also contribute to generated Goodman diagrams, but this function lies outside the main loop.

A function to identify locations of interest for calculating stresses in the presence of strain gages has been developed to offer further verification either to experiment or external simulation results. Currently, this function offers the ability to specify node numbers in the model or gage coordinate locations. The simpler case is if the node numbers in the model are selected as input to the main script, then complex stresses are calculated. The less simple case is if the node locations are specified as input to the main script, then a search routine identifies candidate node locations in the model to find the closest node or set of nodes. This is accomplished by first searching through the list of nodal coordinates in the model and identifying exact matches for all combinations of three Cartesian components down to individual Cartesian components. If no exact matches are found, the routine will search through the list and gather candidate nodes close to specified gage locations within small established tolerances. Once a node is located within a certain tolerance considering all three coordinate directions, complex stresses are calculated. Currently the tolerances, which range from 0.001 to 0.15 to capture a relatively small list of candidate nodes, are hard coded and future iterations of this code should include more robust logic to handle such situations. With all relevant CFD and FEA data extracted and organized, aerostructure calculations including forced response and Goodman diagram generation can proceed.

Code Development: Aerostructure Computations

At this stage, a time history of blade surface unsteady pressures, CFD mesh information, FEA mesh information, modal displacements and stresses, and static stresses are available to be used for forced response analysis. One consideration when using this code is how the model information is organized. It is assumed that the CFD mesh will generally be structured while the FEA mesh will be unstructured. It is also assumed for this case that the CFD mesh will be much coarser than the FEA mesh. The pressures and mode shapes should be resolved to a single mesh in order for the aerostructure interaction to be calculated. Therefore, the mode shape from the FEA mesh is interpolated onto the CFD mesh using a built-in Delaunay triangulation (Ref. 5) of scattered data function in MATLAB®. This interpolant, with the inclusion of a cyclic symmetry scaling factor in Equations (1) to (3), produces interpolated modal displacements represented on the CFD mesh.

$$\vec{u}_{\text{scaled}} = \vec{u}_{\text{unscaled}} * asf \tag{1}$$

$$asf = \sqrt{\frac{nbd}{m}} \tag{2}$$

$$m = \begin{cases} 1; \ if \ \ ND = 0 \text{ or } ND = \dfrac{nbd}{2} \\ 2; \qquad\qquad\qquad \text{otherwise} \end{cases} \tag{3}$$

where $\vec{u}$ is modal displacement, *asf* is ANSYS® scaling factor, *ND* is the nodal diameter analyzed in the current iteration, and *nbd* is the number of blades in the configuration. Given that TURBO outputs unsteady pressures on cell face surfaces, modal displacements are distributed to cell areas via averaging. It is assumed that each cell face will contain four corner nodes with information, so the corner nodes are averaged to represent a modal displacement acting on each cell face. A representative forcing on the blade due to aerodynamic loading and structural response (Ref. 6), referred to here as modal force, is calculated in Equation (4).

$$\text{modal force} = \sum d^2 * P * \left(\vec{A} \cdot \vec{u}_{\text{scaled}}\right) \tag{4}$$

where d is the blade length scaling parameter, P is the nondimensional blade surface unsteady pressure time history, $\vec{A}$ is the CFD mesh cell area vector, and $\vec{u}$ is the modal displacement vector.

A modal force calculation is a scalar product of a force vector with selected eigenmodes that represents by how much each vibration mode contributes to the response of the structure. As this problem includes forced vibration with damping, the solution of the differential Equation (5)

$$m\ddot{x} + c\dot{x} + kx = P_0 \sin \omega t \tag{5}$$

where m is the mass of the oscillator, k is the spring constant, x is the displacement of the oscillator from its equilibrium position, P_0 is the amplitude of the driving force, ω is the angular frequency of the driving force, and t is time, leads to a response solution in Equation (6)

$$x_0 = \frac{\dfrac{P_0}{(2\pi\ \omega_n)^2}}{\sqrt{\left(1-\dfrac{\omega^2}{\omega_n^2}\right)^2+\left(2*\zeta*\dfrac{\omega}{\omega_n}\right)^2}} \tag{6}$$

where x_0 is the response magnitude, P_0 is the magnitude of the modal force harmonic, ω is the forcing frequency, ω_n is the natural frequency, and ζ is the structural damping. Forcing frequency is calculated as a function of rpm, natural frequency is defined as input from ANSYS®, and damping is specified as an input. The modal force harmonic magnitude is calculated by converting the modal force time history into harmonic components using a fast Fourier transform (FFT). The modal force magnitude selected depends on the mode being analyzed. For example, if the mode 1 forced response information is of interest, the harmonic selected from the FFT routine is the first harmonic component, which corresponds to the second entry because the first entry is the average of the frequency domain. Once this calculation is completed, the response is generated as a complex value. This response applied to the modal displacements and stresses effectively converts forces from modal to physical. This step is crucial in determining physical displacements and stresses that the structure undergoes in each operating condition. This directly influences the determination of physical deformation the structure undergoes as well as determining physical stress values for each mode that contribute to high cycle fatigue that can be generally shown in generated Goodman diagrams.

Application and Verification to DTF R24 Geometry in BLI Inflow

The FRAAME code has been developed with the initial goal of performing forced response analysis for the DTF R24 fan blade geometry operating in distorted BLI inflow. The code's structure progresses by handling, in order, unsteady pressures, modal displacements, and modal stresses. It is also written to prioritize scalability. The code can, given proper reference conditions, perform forced response analysis for a general fan blade geometry for any blade in the model, for any nodal diameter, for any single mode or combination of modes. Depending on available information, intermediate visual verification steps are automated as the code progresses. Figure 2 shows mode shapes and stress contours for 100 percent speed, modes 1 through 3, and first nodal diameter to highlight code scalability in generating contours and using information for further calculations.

This report summarizes efforts toward applying the code to the DTF R24 blade geometry for different cases at varying conditions. Case 1 considers 85 percent speed, fourth engine order, mode 2 resonant response. Case 2 considers 100 percent speed, first engine order, modes 1 through 3 nonresonant response modal summation. Results of interest pertaining to the 85 percent speed resonant response case include verification steps in calculating modal force, forced response amplitude, and blade tip displacement, which is compared to the results of Reference 3. Results of interest pertaining to the 100 percent speed, nonresonant response cases include a first engine order, mode 1 (1EOM1) Goodman diagram, which is compared to the results of Reference 4. Case 3: Von Mises stress verification with four strain gages is compared to experiment and the results of Reference 4, and a first engine order, modes 1 through 3 (1EOM1-3) Goodman diagram to demonstrate code-scaling capabilities.

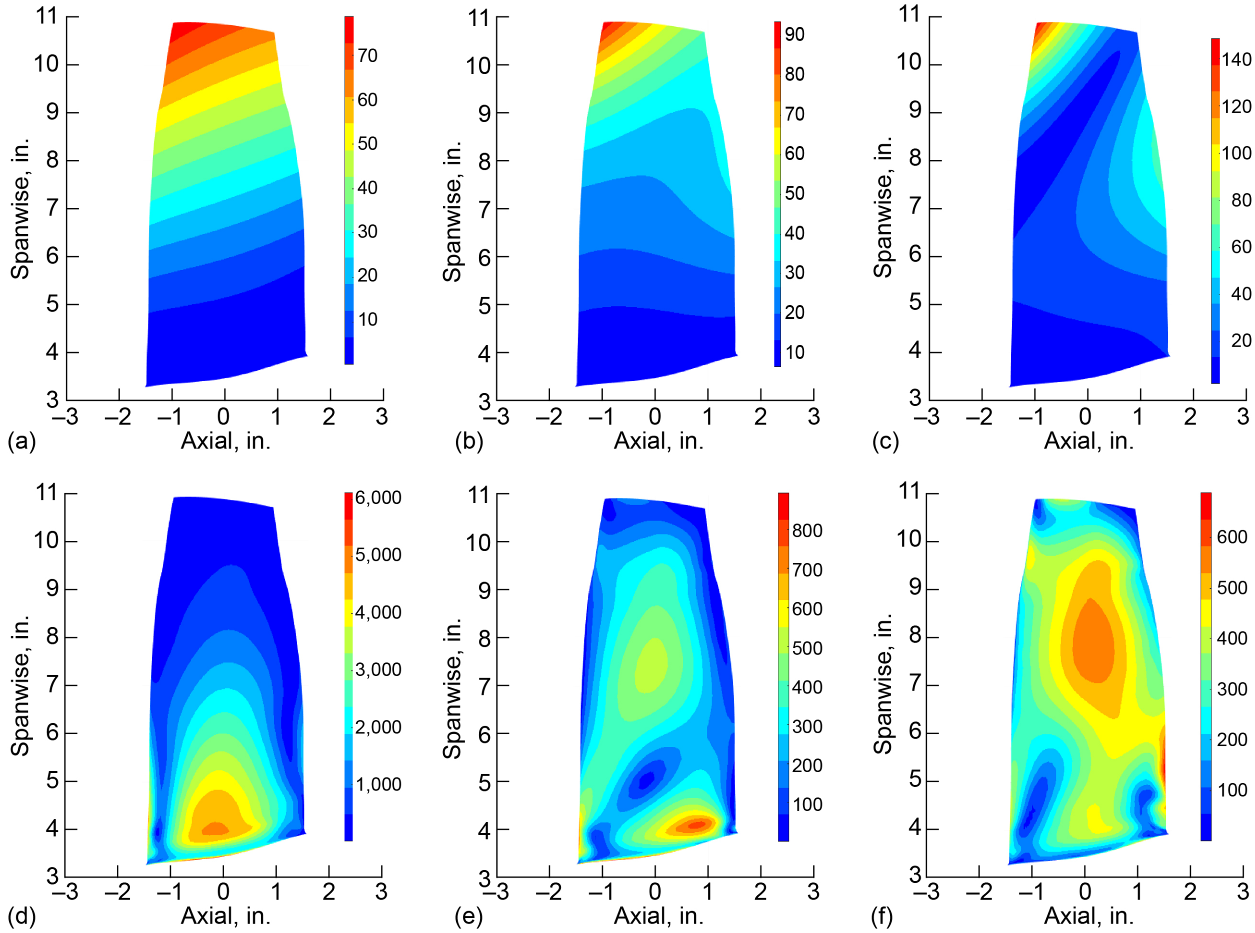

Figure 2.—Mode shapes for modes 1 through 3 and Von Mises stress contour plots for 100 percent (design) speed.

Case 1: 85 Percent Speed, Fourth Engine Order, Mode 2 (4EOM2) Resonant Response

A resonant response can occur if the aerodynamic loading produces an excitation with a frequency close to a mode's natural frequency of a fan blade. For this particular case, a resonant response condition, shown as a crossing of a modal frequency with an engine order excitation frequency on a Campbell diagram, is analyzed at the fourth engine order and second vibration mode at 85 percent rotational speed (Ref. 3). This resonant response can occur as the forcing frequency from the BLI flow approaches the blade's mode 2 natural frequency. Due to this consideration, it is necessary that forced response analysis be performed for this operating condition because it is likely that the fan will approach this speed during operation. Although blade vibration will always occur in some capacity during operation, a resonant response will correspond to one of the worst cases in fan operation. This implies that potentially considerable forcing will affect each blade as the forcing frequency approaches the natural frequency of each vibration mode. Post-analysis efforts can determine if these large forces will accelerate failure due to high cycle fatigue and, in worse cases, sudden catastrophic failure of the structure. This case considers fan operation at sea level, 85 percent rotational speed, and with a structural damping of 1 percent. As this case is intended to isolate a single mode, single engine order, and single blade pressure time history, only one iteration of the previously defined main triple-nested master loop structure is required. Relevant CFD and FEA information is imported and organized, FEA information is interpolated onto the CFD model, and a modal force calculation is performed. At this stage, a frequency filtering routine calculates the forcing frequency as a function of percent speed and mode. This calculated frequency is near the mode 2 natural frequency but is not exactly equal. Because this condition is near a resonant peak, the calculated frequency is discarded and reset to equal the blade natural frequency to represent the worst-case response that the structure can undergo, which implies the largest possible dynamic response. It is also considered that for future analyses, this worst-case condition is of interest, so a check is established in the code to determine if the forcing frequency is within a small enough margin to be considered fully resonant where the frequencies will be set equal. Once the frequencies are determined, the complex forced response amplitude is calculated as a function of modal force, forcing frequency, natural frequency, and structural damping. These parameters are applied, as a magnitude, to the blade tip section to calculate resonant blade tip displacement, which corresponds to the tip probe location of an optical measurement system in the wind tunnel test (Ref. 7). Figure 3 shows a chordwise blade tip displacement plot generated by FRAAME of the physical displacements in mils.

The comparison points in the results of Reference 3 correspond to the blade leading-edge tip displacement of 17.3 mils and a blade tip displacement of 16.2 mils approximately 0.125 in. downstream from the leading edge. Calculations from the FRAAME code include approximate displacement values of 17 and 16 mils in the same locations. This step establishes confidence that the code routines are capable of generating forced response analysis for an isolated mode and nodal diameter case and can be extended to the inclusion of more modes and nodal diameters in a following case study.

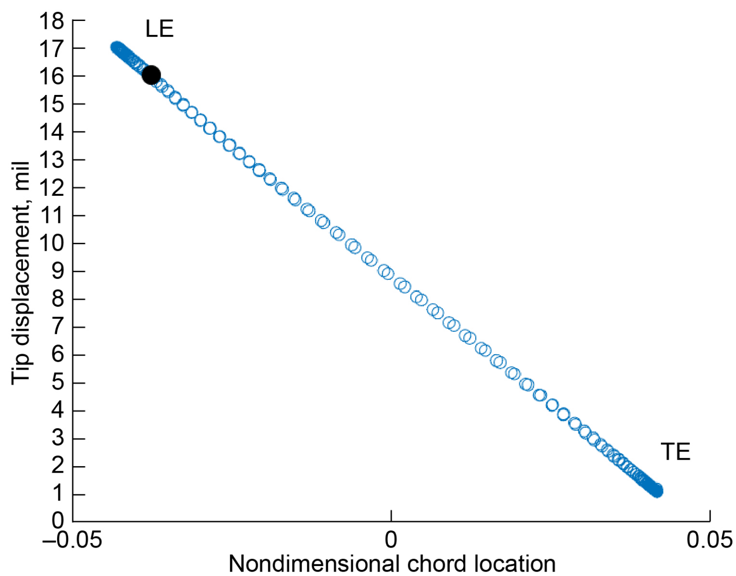

Figure 3.—4EOM2 blade tip displacement in mils.

Case 2: 100 Percent Speed, First Engine Order, Mode 1 (1EOM1) Goodman Diagram

Transitioning between successive cases only requires modification of inputs in the main script as well as compatible ANSYS®/TURBO results abiding by the established file naming scheme. For example, this case considers structural damping of 0.03 percent. Essentially this case follows the same procedure of calculating forced response amplitude with the inclusion of modal stresses. Modal stresses are provided as normal stresses specified at surface nodes in the model. It is worth noting that a special consideration was made with this case as the blade design included various materials for the blade itself (titanium alloy) and disk sector (steel). It is assumed that this unstructured grid will include shared groups of nodes in which modal stresses are averaged for the computation to proceed. The complex forced response amplitude and cyclic symmetry factors are applied to the normal modal stresses and the Von Mises stresses at each surface node are calculated in Equations (7) and (8).

$$\sigma_{VM\text{scaled,complex}} = \sqrt{\frac{\left(\sigma_{xx} - \sigma_{yy}\right)^2 + \left(\sigma_{yy} - \sigma_{zz}\right)^2 + \left(\sigma_{zz} - \sigma_{xx}\right)^2 + 6\left(\sigma_{xy}{}^2 + \sigma_{yz}{}^2 + \sigma_{xz}{}^2\right)}{2}} \tag{7}$$

where stress components on the right-hand side are scaled and complex,

$$\bar{\sigma}_{\text{scaled,complex}} = \bar{\sigma}_{\text{unscaled}} * asf * x_0 \tag{8}$$

where asf is the ANSYS® scaling factor given in Equation (2) and x_0 is the response amplitude given in Equation (6).

Following logic similar to the modal stress extraction routine, the static-only mean stresses are imported and averaged to produce the mean Von Mises stresses. It is worth noting that this particular case includes a duplicate sector where mean stresses are repeated, so a small routine that checks for duplicate information and keeps information from only one sector is included to allow the analysis to proceed. Once the mean and alternating Von Mises stresses are calculated at the surface nodes in the model, a Goodman diagram is generated to display the contribution of mode 1 to the dynamic response of the structure. Figure 4 shows a first engine order excitation with only mode 1 contribution from Reference 4.

Figure 5 shows a similar Goodman plot including a first engine order excitation with only mode 1 contribution generated using similar input to the FRAAME tool.

Although Figure 4 and Figure 5 appear to have minor differences, they show a reasonable degree of agreement, which implies that adequate forced response calculations will produce reasonably accurate Goodman diagram comparisons. It is generally a best practice to consider various modal contributions to the dynamic response of the structure, which is analyzed in this report via modal superposition. Typically, lower modes are excited by lower frequencies, which are encountered more often during operation and contribute more to the dynamic response of the structure than the higher modes. Expanding from a single-mode Goodman diagram, a modal summation, shown in Figure 6, is considered in this report for 100 percent speed, first engine order, and modes 1 through 3 to demonstrate each modal contribution to the dynamic response of the blade. It should be noted that for this version of the MATLAB®-based code, plotting is done manually via the command line.

Figure 6 shows that the mode 1 contribution is the largest contributor to the structural loading for this case.

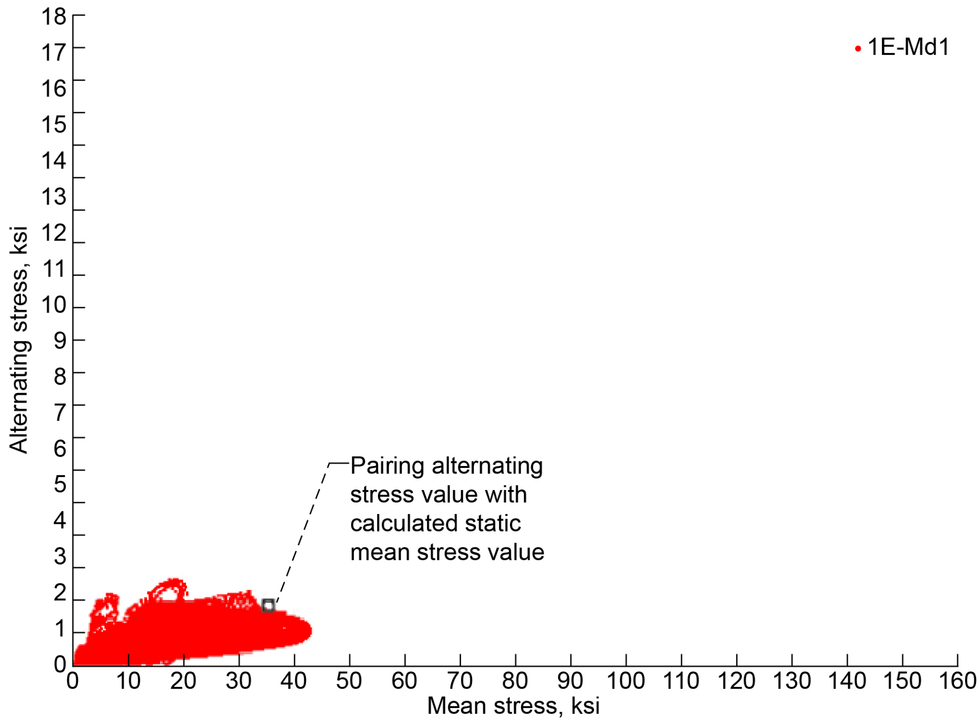

Figure 4.—Goodman diagram: 1EOM1 contribution from Reference 4.

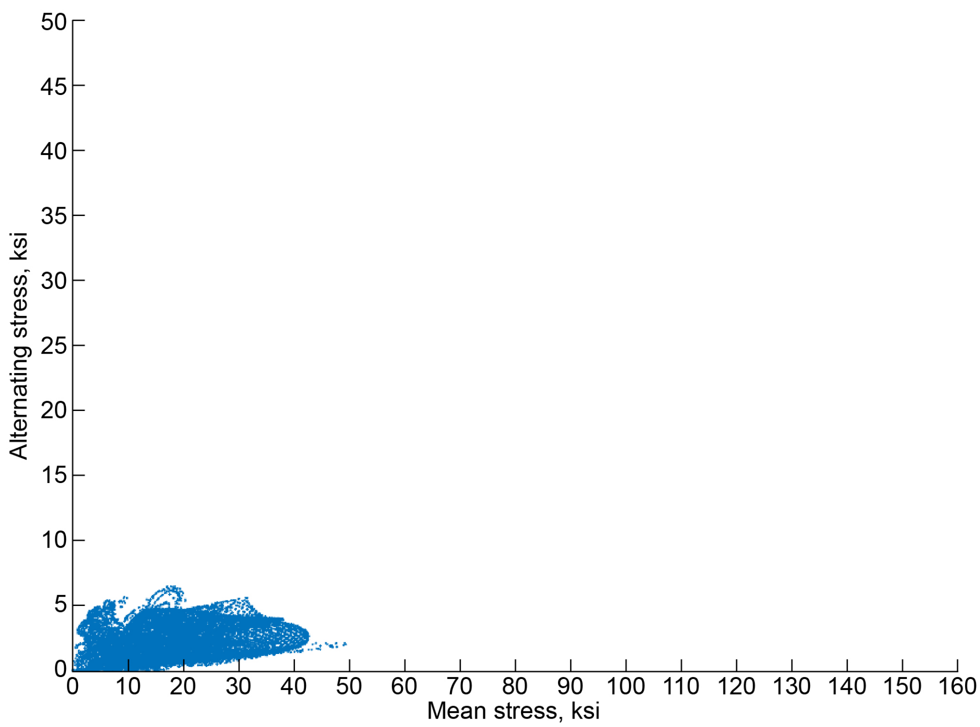

Figure 5.—Goodman diagram: 1EOM1 contribution from FRAAME tool.

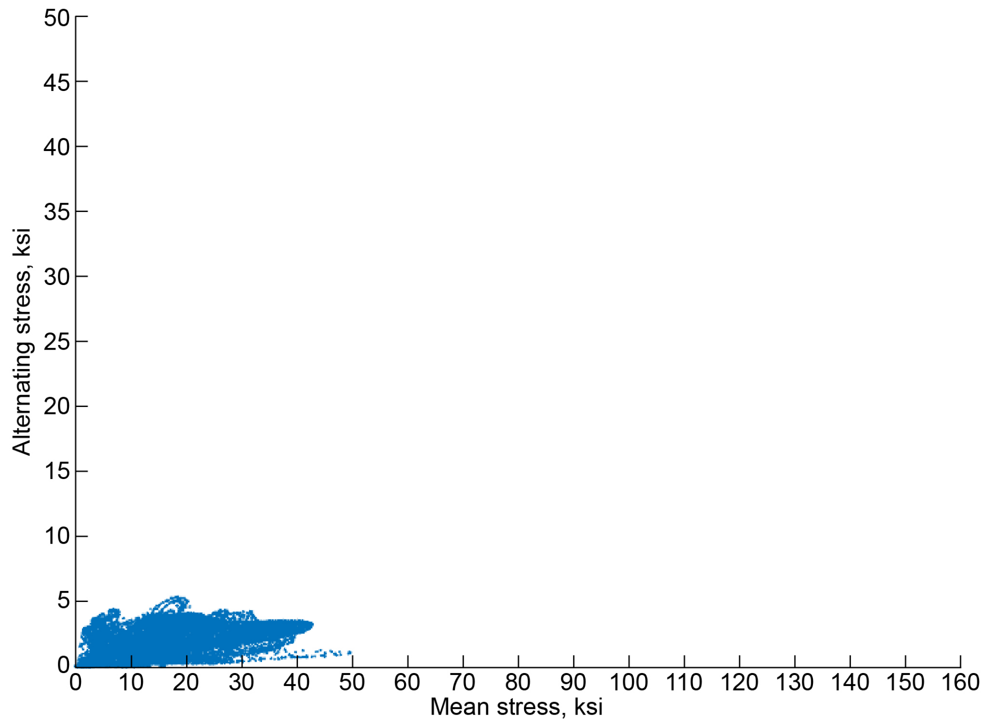

Figure 6.—1EOM1-3 contribution from MATLAB® code.

Case 3: Strain Gage Measurements Versus Prediction

An experimental method of acquiring useful data for structural analysis includes the application of strain gages to test articles to record strains during operation. Strain data for this fan geometry was acquired during a test in the 8- by 6-Foot Supersonic Wind Tunnel facility at NASA Glenn during the BLI^{2}DTF development project (Ref. 8). A preliminary step to acquiring experimental data includes determining the optimal strain gage mounting locations on hardware to capture areas of high dynamic strain during operation. A typical solution involves performing aeromechanics analysis to determine modes and engine orders of interest where potential responses will be detected, as well as utilizing optimization routines for physical positioning of gages. For this particular test, four strain gage locations were determined and provided with physical locations and corresponding computational node numbers for both cold and hot geometries, which correspond to undeformed/manufactured and deformed/operational geometric shapes, respectively. Given that the gages occupy physical space and are not necessarily included in all computational models, the accuracy of the gage location information is not absolute. This final verification step for this code, while requiring information about gage locations, uses basic logic and assumes a compatible physical location is available to match the computational model. The purpose of this step is to generate a method of using the structural model for each mode and engine order to perform forced response analysis and determine a geometric location in which calculated stresses may be correlated to experimentally measured strains. This routine, embedded in the main loop of the code, includes a basic searching function that, given either strain gage coordinate information or node locations in the model, calculates the physical stress components and Von Mises stresses at each identified location at which a gage is expected to exist.

The caveat with this procedure is that the availability of strain gage information is the most critical step. For example, given various gage locations, the computational model may not necessarily share a node close to the prescribed location on the blade. Experimental data is collected with the loaded hot geometry while the gage installation process occurs with the unloaded cold geometry. Computational models can be generated for both hot and cold shapes with varying locations of nodes due to the geometric deformation caused by loading. A potential resolution to this issue includes correlating hot and cold geometries to account for model differences in simulation and experiment. It should be noted for this effort that the hot and cold shape model differences are not considered, so effects of changing gage locations, including gage orientation, are neglected. A manual input will allow the user to determine and enter the information on both gage locations in Cartesian coordinates and gage node numbers in the model, but only one will be used for analysis. This is to prevent one input from taking priority over the other in the absence of complete information on gage location and node number. If a gage location is specified, a routine in the code searches through various Cartesian coordinates to determine various node numbers with values similar to the requested location within a small tolerance. Currently, these tolerances can be modified to expand or limit nodes captured by the routine to provide options if more than one node near the location of interest is requested. Too wide of a tolerance will produce many results and too narrow of a tolerance will produce few to no results. Future iterations of the FRAAME code will explore expanded logic to make this process more robust in searching for compatible node locations.

The current logic of searching through the model coordinates prioritizes them in this order: y, x, and z. In other words, the y coordinates, which correspond to the spanwise location of a gage or the location with the largest assumed variations in gage location due to blade design proportions, are searched for first. Next, the x coordinates are searched, followed by the z coordinates. This candidate selection process implies that the full model is searched for y coordinates, the filtered candidates are searched for x coordinates, and the final filtered candidates are searched for z coordinates. Through trial and error, this methodology produces a small batch of candidate nodes to be considered. In this case, exactly one candidate node for each gage was returned by this selection routine. Once the node is selected, modal stress information based on the node number in the model is extracted, scaled with forced response results, and physical stress components and Von Mises stresses are calculated. This information corresponds to the calculated stress at a location that should theoretically be correlated to the physical location of a strain gage. It is assumed for this analysis that strain gage correlation with experiment and FEA was performed in a previous study (Ref. 4). Another feature of this routine is a much simpler process in which gage node numbers in the model are prescribed or known. If the node number is known, the modal stress information is extracted directly from the corresponding model and physical stresses and Von Mises stresses are calculated. This analysis uses both methods in the routine including analyzing gage locations and node numbers. Validation for this case includes calculating complex stress components for each gage at the first engine order for modes 1 through 3. These components are summed to represent modal contributions and Von Mises stresses are calculated as a representation of the measured strains, then stresses, at each gage through this operating regime. The results are compared to a previous study in Reference 4 in which contributions of many modes, more than available for this analysis, produce a numerically converged solution for forced response analysis to be compared to experimental data shown in Figure 7.

Figure 8 shows a similar plot generated by FRAAME at the first data point of comparison for the first three modes.

It should be noted that this effort only includes information requested for 100 percent speed, first engine order, modes 1 through 3. The primary comparison point for Figure 7 and Figure 8 includes a three-mode summation; further mode summations are unavailable. It should also be noted that the code architecture, prioritizing scalability, is built to handle as much or as little information as is provided. In theory, information included in the compared study (Ref. 4) can be included here as long as the appropriate modal files are included in the working directory and proper nomenclature is maintained.

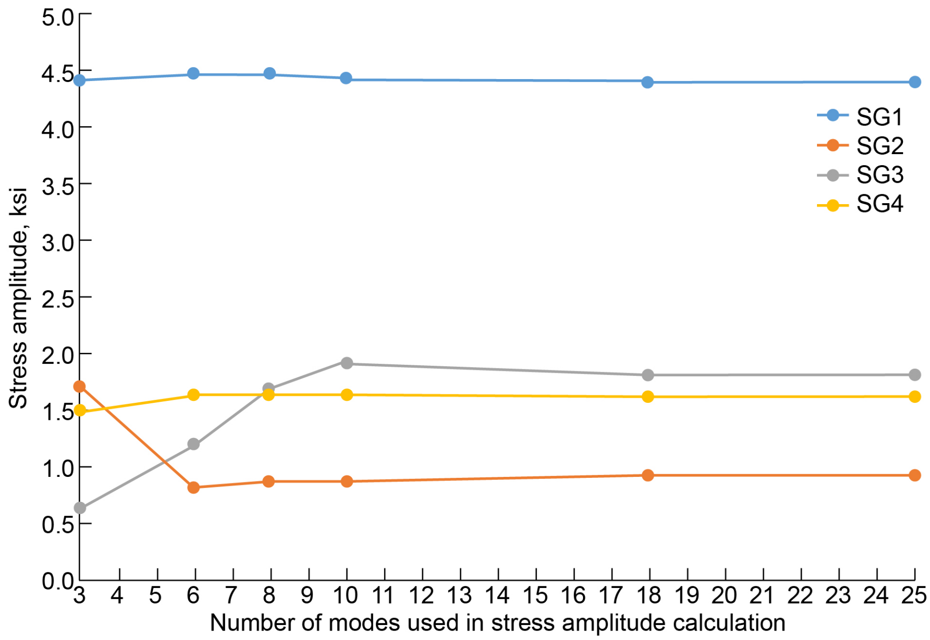

Figure 7.—Modal summation convergence study for strain gages (SGs).

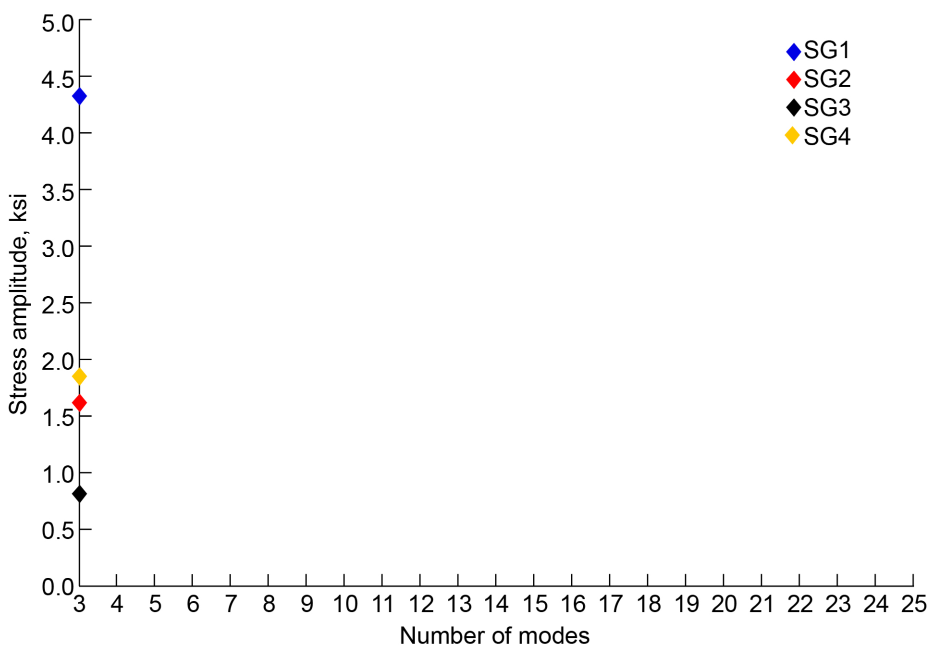

Figure 8.—First three mode summation for strain gages (SGs) generated by FRAAME.

Concluding Remarks

This report describes the development and application of the research-level Forced Response Aeromechanics Analysis in a MATLAB-Based Environment (FRAAME, Version 1) tool developed at NASA Glenn Research Center that seeks to generate a streamlined forced response analysis framework with inputs from the TURBO computational fluid dynamics (CFD) code and finite element analysis (FEA) solver ANSYS® (Ansys, Inc.). The FRAAME tool is written in a functional, modular form and with minimal hard-coded inputs to be user friendly and versatile in its use and modification. It also includes various plotting stages for information that can be considered useful, such as contour plots and Goodman diagrams. The code's looping structure allows including further information, such as blade surface pressure for different blades in the annulus, more modes, and more engine orders, depending on information availability. Once a generalized response amplitude is calculated for each engine order, the code can perform a modal summation routine and generate a Goodman diagram to demonstrate each mode's contribution to the high cycle fatigue of the structure through various operating conditions. This effort is supplemented by producing results that are validated with results from previous studies pertaining to the R24 geometry with the same reference conditions and CFD/FEA results used here as input. Unsteady aerodynamics results from TURBO and structural dynamics results from ANSYS®, both generated for previous analyses referenced, are used to calculate forced response and high cycle fatigue. This type of analysis may not only contribute to smarter designs of turbomachine structures but also assists in tool validation when a compatible real-world experiment can be referenced.

The first version of the code is designed specifically to analyze the R24 geometry from the Boundary Layer Ingesting Inlet/Distortion-Tolerant Fan project, but it includes various considerations to extend its use to other proposed fan and compressor designs. Due to these considerations, minimal hard-coded inputs are included to produce a mostly autonomous, generalized forced response and high cycle fatigue analysis framework. Future improvements to FRAAME will include smarter logic for identifying strain gage locations based on coordinates, porting of the code to a central location such as the NASA Advanced Supercomputing system for large-scale analyses, improvements to areas such as plotting tools and graphical user interfaces, investigation of the inclusion of various additional external CFD/FEA codes, and implementing analysis capabilities for mistuned systems.

References

1. Chen, Jen Ping; and Whitfield, David L.: Navier-Stokes Calculations for the Unsteady Flowfield of Turbomachinery. AIAA 1993–0676, 1993.
2. Chen, J.-P.; and Briley, W.R.: A Parallel Flow Solver for Unsteady Multiple Blade Row Turbomachinery Simulations. ASME GT2001–0348, 2001.
3. Bakhle, Milind A., et al.: Aeromechanics Analysis of a Distortion-Tolerant Fan With Boundary Layer Ingestion. AIAA SciTech Forum, Kissimmee, FL, 2018.
4. Min, J.B., et al.: Cyclic Symmetry Finite Element Forced Response Analysis of a Distortion-Tolerant Fan With Boundary Layer Ingestion. AIAA SciTech Forum, Kissimmee, FL, 2018.
5. Amidror, Isaac: Scattered Data Interpolation Methods for Electronic Imaging Systems: A Survey. J. Electron. Imaging, vol. 11, no. 2, 2002, pp. 157–176.
6. Den Hartog, J.P.: Mechanical Vibrations. Dover Publications, 1985.
7. Duffy, Kirsten P., et al.: Laser Displacement Measurements of Fan Blades in Resonance and Flutter During the Boundary Layer Ingesting Inlet and Distortion-Tolerant Fan Test. AIAA SciTech Forum, Kissimmee, FL, 2018.
8. Provenza, Andrew J.; Duffy, Kirsten P.; and Bakhle, Milind A.: Aeromechanical Response of a Distortion-Tolerant Boundary Layer Ingesting Fan. ASME J. Eng. Gas Turbines Power, vol. 141, no. GRP–18–1308, 2018.