NASA/TM—2000-209635

Unsteady Cascade Aerodynamic Response Using a Multiphysics Simulation Code

C. Lawrence
Glenn Research Center, Cleveland, Ohio

E. Spyropoulos
ANSYS, Inc., Sunnyvale, California

T.S.R. Reddy
Univeristy of Toledo, Toledo, Ohio

January 2000

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's 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.
- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored 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 data bases, 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 Access Help Desk at (301) 621-0134
- Telephone the NASA Access Help Desk at (301) 621-0390
- Write to:
  NASA Access Help Desk
  NASA Center for AeroSpace Information
  7121 Standard Drive
  Hanover, MD 21076

NASA/TM—2000-209635

Unsteady Cascade Aerodynamic Response Using a Multiphysics Simulation Code

C. Lawrence
Glenn Research Center, Cleveland, Ohio

E. Spyropoulos
ANSYS, Inc., Sunnyvale, California

T.S.R. Reddy
Univeristy of Toledo, Toledo, Ohio

National Aeronautics and
Space Administration

Glenn Research Center

January 2000

Acknowledgments

The NASA High Performance Computing and Communications Program supported this work.

Available from

NASA Center for Aerospace Information
7121 Standard Drive
Hanover, MD 21076
Price Code: A03

National Technical Information Service
5285 Port Royal Road
Springfield, VA 22100
Price Code: A03

Unsteady Cascade Aerodynamic Response Using a Multiphysics Simulation Code

C. Lawrence
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

E. Spyropoulos
ANSYS, Inc.
Sunnyvale, California

T.S. R. Reddy
University of Toledo
Toledo, Ohio

Abstract

The multiphysics code *Spectrum*™ is applied to calculate the unsteady aerodynamic pressures of oscillating cascade of airfoils representing a blade row of a turbomachinery component. Multiphysics simulation is based on a single computational framework for the modeling of multiple interacting physical phenomena, in the present case being between fluids and structures. Interaction constraints are enforced in a fully coupled manner using the augmented-Lagrangian method. The arbitrary Lagrangian-Eulerian method is utilized to account for deformable fluid domains resulting from blade motions. Unsteady pressures are calculated for a cascade designated as the tenth standard, and undergoing plunging and pitching oscillations. The predicted unsteady pressures are compared with those obtained from an unsteady Euler code referred in the literature. The *Spectrum*™ code predictions showed good correlation for the cases considered.

Introduction

Historically, the broad scope of aeropropulsion multidisciplinary applications necessitates that a collection of approaches, with distinct capabilities, be developed (Ref. 1). Several classes of multidisciplinary tools are currently employed in the aeropropulsion industry. The "Loosely coupled" approach, where existing single disciplinary codes are run, data is generated, made available and used for subsequent analysis, is the most widely approach in use today. This approach is implemented in industry by having separate disciplinary "departments" where one department (e.g. aerodynamics) generates results and passes these results on to subsequent departments (e.g. structures). For this approach, the data must be in the correct format for use by the subsequent analysis, but the subsequent code need not directly communicate with the previous code. The limitation of this approach is that the physics of the problem must be loosely coupled or incorrect results may be produced.

A second approach of coupling is the "coupled process" approach where individual disciplinary codes are used, similar to the loosely coupled approach. However, in the coupled process approach the disciplinary codes are linked and run concurrently with each other. An example of this approach is coupling an engine inlet aerodynamic code with a compressor code to predict the system response on an inlet coupled to the engine compressor. A major disadvantage of this approach is that the capabilities of the coupled codes tend to be application specific and additional effort is normally required to extend the capabilities to other applications.

The third approach is the "system of equation-fully coupled approach", or multiphysics approach, where a single code is used and all the disciplines are represented in the fundamental equations. This approach addresses those applications whose characteristics require that the disciplines be coupled at the fundamental equation level to accurately, and/or more efficiently, capture the multidisciplinary physics of the problem. The coupled process approach may be applicable to a limited number of such applications, however, the coupled process approach requires that specialized tools be developed to exchange and manage data among disciplinary codes. The multiphysics approach overcomes the limitations of the previously described approaches by providing a single code accurately containing all the necessary disciplines thus eliminating the need for data exchange, data exchange tools and the overhead associated with maintaining a collection of separate disciplinary codes.

NASA Glenn Research Center initiated a project to evaluate the *Spectrum*™ code for multidisciplinary aeropropulsion applications. As part of this project a collection of in-house and industry selected test cases were used to assess the potential use of this code as an industry tool for multidisciplinary simulations. *Spectrum*™, a product developed by Centric Engineering Systems, Inc. (Refs. 2 and 3), currently ANSYS, Inc., provides the ability to include fluids, structures, and their interactions with each other in a single-pass simulation. In this simulation, the engineer can capture the true multiphysics behavior of products. Furthermore, this is accomplished in only one software program and in a single simulation run. Instead of a collection of existing codes, *Spectrum's*™ multiphysics capabilities are provided within the fundamental equations used to characterize the problem. Currently, there are no other available commercial products that possess the broad scope of multiphysics capabilities available in *Spectrum*™. As a part of this project, results for four aeropropulsion related problems are currently available; (1) a disk quenching simulation, (2) a drum rotor problem, (3) a centrifugal compressor simulation, and (4) a cooled cowl lip. Results from these simulations are available in Ref. 7 and more detailed reports are available from Centric Engineering Systems, Inc.

Aeromechanical Applications for Turbomachines

Two important multidisciplinary aeropropulsion problems are flutter and forced response. These phenomena occur as a result of interactions between the aerodynamic flow and the structural behavior of the turbomachinery blading. Forced vibrations may lead to high cycle fatigue failures, while flutter is a self-excited event that may lead to unstable aeromechanical situations. Numerous computational techniques are available for predicting flutter and forced response (Ref. 11). Computer codes are available for two and three-dimensional aeromechanical simulations, bladed disks and cascades, subsonic through supersonic flow, and frequency and transient analysis. Unfortunately, most of these techniques are designed for specific aerodynamic regimes, mechanical configurations, or types of aeromechanical behaviors and therefore often require modifications before they may be applied to other types of aerodynamic or mechanical conditions. To overcome the limitations of application specific aeromechanical techniques, and to minimize the costs associated with developing and maintaining a collection of codes, general-purpose multidisciplinary codes that are applicable to a broad class of aeropropulsion, as well as general multidisciplinary problems, must be developed and employed. There are two major benefits to implementing a general-purpose simulation code. First, these codes are able to more accurately capture the physics by integrally modeling the fluids and structures through the basic physical equations, and second, the same model can be used for multiple purposes. For example, instead of using one set of codes to predict flutter then another set of codes for aerodynamic performance and still another for general structural design, a single model and code can be used for all aspects of the design. A disadvantage of general-purpose codes is that they often are computationally demanding and often require considerable more run time than specialized simulation codes which can be computationally optimized for their specific application.

An overall requirement for a general-purpose aeromechanical simulation tool is the ability to perform transient simulations. Having transient analysis capabilities enables frequency based analysis to be done, as well, since frequency data can be extracted from the transient results. Also, a transient simulation can be used to model aerodynamic and structural nonlinearities (e.g. aerodynamic airfoil loads are nonlinear for large blade motions). The disadvantage of this approach is that very long response duration are required leading to simulations that may take weeks or even months to run.

In the area of modeling, there is a requirement for general three-dimensional capabilities to model the structural and aerodynamic characteristics of real blade geometries. There is also a need for a fully coupled aerodynamic and structural capability to capture the true interactions between the flow field and the structure. Most capabilities in use today are based on some form of assumed structural motion that is used to predict the resulting aerodynamic response. The aerodynamic response then is used to assess the validity of the assumed structural motion. This approach may be adequate for most applications, however, it may lead to an omission of certain types of aeromechanical response.

To capture all possible aeromechanical responses either a coupled aero-structural simulation where aero and structural state variables are exchanged at least every time step or a fully coupled simulation, where all motions are derived from the physical system, may be required. A very limited number of works address the fully coupled simulation. The most relevant to this project was performed by Bendikson (Ref. 14) where flutter calculations were performed for a cascade using a coupled fluid-structure system. This work was limited to two-dimensional systems and structures that can be modeled using plate finite elements. The *Spectrum*™ code has also been used for studying panel flutter and wing flutter by Rifai, et al, Ref. 7 and for aeromechanical study of airfoils undergoing pitching and plunging oscillations by Grisval, et al, Ref. 8. They showed good correlation with published results. However, the study was restricted to isolated wings and airfoils. To the authors' knowledge, the *Spectrum*™ code is the only commercial code providing a fully coupled simulation capability with three-dimensional modeling and a robust selection of finite elements and fluid and structure material models.

Model Description and Analysis Approach

In the present study the *Spectrum*™ code is applied for the prediction of aerodynamic unsteady blade loading of oscillating cascade of airfoils representing a blade row of a turbomachinery component. The predicted unsteady loading can be used in a post stability analysis for prediction of flutter and forced response. The calculated results will be compared with those obtained from, LINFLX2D code, Ref. 9, developed at NASA Glenn Research Center. The LINFL2D code is a frequency domain based linearized unsteady Euler cascade solver. The unsteady equations were linearized about a steady solution. The steady solution is obtained from a non-linear Euler solver, NPHASE, Ref. 10. Both of these NASA codes solve the fluid mechanics equations on structured meshes using a combined flux differencing and flux splitting numerical solution scheme. In both codes, the structure and aerodynamic solvers are loosely coupled. The NASA codes to date have been validated only for problems for which the deformations are within linear range.

The *Spectrum*™ code is a multiphysics simulation software package based on the finite element method (Ref. 2). The code includes compressible and incompressible fluid flow, structural, and thermal modeling as well as the interactions between these disciplines. Interaction among disciplinary domains is enforced using the augmented-Lagrangian method. The finite element representation of the fluids is based on the Galerkin-Least-Squares (GLS) method and the arbitrary-Lagrangian-Eulerian (ALE) method is used to account for deformable fluid domains. Reynolds-averaged and large-eddy simulation models are utilized for turbulence simulation. In this treatment, the compressible flow formulation makes use of physical entropy variables (Ref. 2). With these variables, the fluid conservation laws are expressed in symmetric form that intrinsically expresses the mathematical and physical stability provided by the second law of thermodynamics. In turn, *Spectrum's*™ finite element formulation inherit this fundamental stability and convergence proofs are available (Ref. 3). The finite element characterization of the structural domains employs a 3-field formulation based on the Hu-Washizu variational principle (Ref. 4). This method is well documented in the literature to address numerical locking phenomena (Ref. 5). The kinematic description admits small and finite deformations and strains. Linear and nonlinear material models are used for the constitutive relations with thermo-mechanical coupling. The multiphysics approach implemented in *Spectrum*™ is based on a model that is defined as a hierarchical tree of regions and interfaces. Regions are used to define a physical characteristic of a domain, such as fluid or solid behavior, then interfaces are used to enforce coupling between regions. The *Spectrum*™ software is available on both serial and parallel computing systems.

Figure 1 depicts the modeling, simulation and post-processing procedure followed in the current work. Geometry and mesh generation was performed using PATRAN, a widely used commercial pre- and post-processing package. Both fluid and solid domains were modeled and meshed using

PATRAN. Meshing of the domains was based on sufficiently capturing the fluid and solid behavior without any additional or unique requirements due to the coupling between fluid and solid domains or the multidisciplinary nature of the simulation. Once the domain meshes are created, the *Spectrum Editor* was used to extract boundary condition information. Files containing lists of elements and nodes on the fluid boundaries, and the boundaries between fluid and solid domains, were obtained using the *Editor*. Use of the *Editor* is necessary since PATRAN, which is primarily a pre-processor for solid models, is not designed to readily extract this information and to put it into a format acceptable for the *Spectrum Solver*. After the analysis model was completed, the *Spectrum Solver* was used to perform the actual simulation and to generate visual and numerical transient fluid and solid output.

The *Spectrum Visualizer* was used for visualization and a specially written post-processor was used to process the output data into a format for making comparisons with published aeromechanical data. After transient simulation data was computed using the *Spectrum*™ code, several additional steps are required to render the data in a format that was appropriate for comparison to the aeromechanical data available in the literature. The first step was to extract the pressures as a function of time on the airfoil pressure and suction surfaces and convert them to the frequency domain using a Discrete Fourier Transform (DFT). At least four vibration cycles were used in this step. Next, the values of the DFT at the driving frequency were extracted and the difference between the airfoil pressure and suction surface values were normalized and plotted as a function of airfoil chord distance.

The aeromechanical model used for this study consists of two typical section airfoils in a fluid domain (Figure 2). The airfoil geometry is the tenth standard configuration obtained from Ref. 12. This configuration is a modified cambered NACA 0006 airfoil. This configuration was chosen so a comparison could be made between the results of the present simulation and already validated and industry accepted results. Two blade passages are modeled so that out of phase blade motions could be analyzed. The inlet and exit boundaries were placed 1 chord length away for direct comparison with the published results.

The fluid was modeled using air as an ideal compressible inviscid gas. The boundary conditions at the airfoil surface were modeled as no mass flow or velocity normal to these surfaces. At the inflow, the total pressure and total temperature were specified. The flow angle was also defined at the inlet. The flow velocity and mesh displacement in the spanwise z-direction was set to zero since the actual problem is two-dimensional. The static pressure was set at the outflow. Initial pressure, temperature and velocity were specified throughout the fluid domain using results obtained from a steady state fluid only simulation (see below). Nodes on the top and bottom of the upper fluid region were tied to opposite nodes on the bottom and top of the lower region, to model multiple blade passages, using periodic boundary conditions.

The finite element mesh is shown in Figure 3. Two fluid domains, corresponding to the upper and lower blade passages, were modeled. Identical meshes are used for the upper and lower fluid regions and solid regions. Since *Spectrum*™ is a 3-D code, a 3-D grid was constructed that had only one element along the spanwise direction. Each of the fluid domains is modeled with 15702 wedge elements. Considerable mesh refinement is provided at the airfoil leading and trailing edges where there is predicted to be relatively large variations in the fluid flow parameters. The solid domains are modeled using 243 hexahedral elements. A relatively coarse mesh is used for the airfoil solid domains since airfoil elasticity is neglected and instead, the airfoils are caused to move by applying enforced pitching and plunging displacements. It is important to note that it is not required that the fluid and solid meshes match at their interfaces. This unique feature of the *Spectrum Solver* allows for modeling coarseness or refinement based solely on the physical behavior of the domains rather than on the geometric coupling between domains. The generality of the interface treatment permits a variety of interaction constraints to be used independently on the mechanical, thermal and mesh field variables. A slave-master algorithm is used to impose continuity relations between two sides of an interface. The interaction constrains are enforced in a fully coupled manner (Ref. 2).

All of the simulations were performed using the nonlinear dynamic capability of *Spectrum*™. The structural response and the coupling between the structure and fluid were chosen to be characterized with linear behavior, however, a nonlinear solution was required for the fluid domain. Simulations were performed using both the serial and parallel versions of *Spectrum*™ depending on the availability of computational resources. The speed-up with the parallel simulations was approximately scalable with the number of processors used.

Two solid domains, corresponding to the upper and lower airfoils, were modeled. The solid was modeled using a small deformation, linear material model. Displacements at the airfoil leading and trailing edge nodes were applied to model the prescribed pitching and plunging motions. The actual time histories of these motions were computed from the amplitudes and reduced frequency specified in Ref. 12 and given in Table 1.

For the fluid only steady state simulation, local time stepping was employed. This technique enables small time steps to be applied to the regions in the fluid, which is modeled with a finer mesh (i.e. smaller element sizes), and larger time steps where the mesh is coarse. Utilization of this capability considerably reduces the computational time to reach steady state without any degradation in the fidelity of the simulation results. Further efficiency is provided by applying different time integration parameters for fluid, structure and ALE mesh used to couple fluid to structure. The time increment was set to 3.5×10^{-5} seconds in the transient simulations.

As noted above, to minimize the computational effort required for the transient simulations, a steady state solution was run without any airfoil motion at each Mach number to obtain the steady flow condition then the results from the steady solution were used as initial conditions for the transient simulation. To obtain the steady solution several runs were required where the exit pressure was varied until the desired inlet Mach number was obtained. This was required since only the inlet Mach number value was reported in Ref. 12. Once this combination of boundary conditions was found the flow field temperatures, pressures and velocities were used as initial conditions for the transient simulations. The present model allows for the airfoil motions to be out of phase with each other, however, for this study only in-phase airfoil motions were used.

Results

Calculations were made for a cascade comprising of airfoils designated as the tenth standard configuration (C10). The cascade has a gap to chord ratio of 1.0 and a stagger angle of 45 degrees. The C10 airfoils are constructed by superposing the thickness distribution of a modified NACA 5506 airfoil on a circular arc camber line. See Ref. 12 for more details. Simulations were performed for two Mach numbers; Mach 0.70 where the flow is entirely subsonic and Mach 0.80 where the flow is transonic with a shock in each blade passage. The steady angle of attack is 10 degrees for M=0.7 and thirteen degrees for M=0.8. A grid refinement study was performed to ensure that the grid resolution was adequate. There are 80 points on the airfoil. The *Spectrum*™ code was run in inviscid mode. Table 1 shows the combination of simulations that were performed for this study.

Steady Flow Calculations:
The steady Mach number distribution plotted as a function of chord distance for M = 0.7 and M = 0.8 is shown in Figures 4a and 4b, respectively. For Mach 0.70 the steady angle of attack is 10 degrees and the flow is entirely subsonic. For Mach 0.80 the steady angle of attack is 13 degrees and the flow is transonic with a normal shock occurring in each blade passage. The steady Mach number distribution obtained from NPHASE solver, Ref. 10, is also shown in Figure 4. For both Mach numbers, the results agree well, except that the Spectrum solver predicts the Mach 0.80 shock location slightly downstream of that predicted by the NPHASE solver. This may be due to the different grid size and distribution. However, the overall qualitative and quantitative trends are the same.

Unsteady Flow Calculations:
Unsteady pressures are calculated for blades oscillating in pitching and plunging. The calculations were made for zero interblade phase angle ($\sigma = 0^\circ$) motion. As noted earlier, out of phase motions ($\sigma = 180^\circ$) are also possible with the present setup.

Figures 5 and 6 show the unsteady pressure difference coefficient distribution for M=0.7 for pitching and plunging motion. The pitching motion is about mid chord. For this Mach number the flow is shock free.

The unsteady pressure difference coefficient, ΔC_P, is obtained by normalizing the pressure difference between the pressure and suction surfaces. In formula:

$$\Delta C_P = (P_{\text{suction}} - P_{\text{pressure}}) \,/\, ((\rho * U^2 * \text{abs}(\alpha_o + h_0 * \overline{\omega}))$$

Where P_{suction} is the pressure on the suction surface, P_{pressure} is the pressure on the pressure surface, ρ is the air density, U is the free stream velocity, α_o is the amplitude of pitching motion, h_0 is the amplitude of plunging motion, and $\overline{\omega}$ is the reduced frequency based on chord. The blades are oscillated at a reduced frequency, $\overline{\omega}$, of 1.0. The reduced frequency is defined as $\overline{\omega} = \omega * c / U$, where ω is the vibration frequency in radians, c is the chord and U is the free stream velocity. The results are compared with those obtained from LINFLX2D solver, Ref. 9. As mentioned earlier, LINFLX2D is a linearized unsteady aerodynamic solver based on the steady solution from NPHASE. However, with LINFLX2D the frequency domain unsteady aerodynamic coefficients are directly obtained in the solution process. As mentioned earlier with *Spectrum*™, the time transients are calculated first, and then they are Fourier decomposed to obtain the unsteady aerodynamic coefficients. Good agreement between the *Spectrum*™ results and LINFLX2D is seen in both the cases. The agreement is especially good at the airfoil leading edge where high loading is expected.

The unsteady pressure difference coefficient for M=0.8 is shown in Figures 7 and 8. Figure 7 shows for pitching motion and Figure 8 for plunging motion. Again the results are compared with those obtained from LINFLX2D. Similar to the results for M=0.7, they show good agreement. There is some difference at or near the shock location, which was the consequence of predicting the shock location at slightly different location. The transonic results are very sensitive to grid size and distribution, and for comparison one has to use the same grid size and distribution. Despite of this, the overall trends are the same.

Conclusion

The multiphysics simulation code, *Spectrum*™, was applied to predict unsteady aerodynamic response of oscillating cascades in pitching and plunging motion. The oscillating cascades represent a blade row of a turbomachinery component. The analysis is based on the finite element method, and the approach is based on a single computational framework for the modeling of multiple interacting physical phenomena. The augmented-Lagrangian method is used to enforce interaction constraints among all field variables in a fully consistent manner. The arbitrary-Lagrangian-Eulerian method is utilized to account for deformable fluid domains. The predicted unsteady pressures showed good agreement with those obtained from an unsteady Euler code. Since the *Spectrum*™ code is basically a three-dimensional code, aeromechanical simulation of the three-dimensional turbomachinery rotor and stators is straightforward. Future applications include the extension of this study to other interblade phase angles, prediction of gust response of a turbine cascade, and multiple blade row simulation.

References

1. Lawrence, C., “An Overview of Three Approaches to Multidisciplinary Aeropropulsion Simulations”, NASA/TM–107443, May 1997.

2. Spectrum Theory Manual, Centric Engineering Systems, Santa Clara, California, 1994.

3. Spectrum Users Manual, Centric Engineering Systems, Santa Clara, California, 1994.

4. Johnson, C., Szepessy, A., and Hansbo, P., "On the Convergence of Shock-capturing Streamline Diffusion Finite Element Methods for Hyperbolic Conservation Laws," Mathematics of Computation, Vol. 54, pp. 107–129, 1990.

5. Simo, J.C. and Rifai, M.S., "A Class of Mixed Assumed Strain Methods and The Method of Incompatible Modes," International Journal for Numerical Methods in Engineering, Vol. 29, pp. 1595–1638, 1990.

6. Simo, J.C., Taylor, R.L., and Pister, K.S., "Variational and projection Methods for the Volume Constraint in Finite Deformation Elastoplasticity," Comp. Meth. App. Mech. and Eng., Vol. 51, 1985.

7. Rifai, S.M., Ferencz, R.M., Wang, W., Spyropoulos, E.T., Lawrence, C., and Melis, M.E., "The Role of Multiphysics Simulation in Multidisciplinary Analysis", NASA/TM—1998-208653, October 1998.

8. Grisval, J., Sauvignet, C., and Johan, Z., "Application of the Finite Element Method to Aeroelasticity", September 1996.

9. Verdon, J.V., Montogomery, M.D., and Kousen, K.A., "Development of a Linearized Unsteady Euler Analysis for Turbomachinery Blade Rows", NASA CR–4677, June 1995.

10. Swafford, T.W., et al, "The Evolution of NPHASE: Euler / Navier-Stokes Computations of Unsteady Two Dimensional Cascade Flow Fields", AIAA Paper 94–1834, 12th Applied Aerodynamics Conference, Colorado Springs, Colorado, June 20–23, 1994.

11. Reddy, T.S.R., et al , "A Review of Recent Aeroelastic Analysis Methods for Propulsion at NASA Lewis Research Center", NASA TP–3406, December 1993.

12. Atassi, H.M., "Unsteady Aerodynamics, Aeroacoustics, and Aeroelasticity of Turbomachines and Propellers, Springer-Verlag, 1993.

13. Bakhle, M.A., et al, Development of an Aeroelastic Code based on an Euler / Navier-Stokes Aerodynamic Solver, 96-GT-311, International Gas Turbine and Aeroengine Congress & Exhibition, Birmingham, UK, June 10–13, 1996.

14. Bendikson, O.O. and Hsiao, C., "New Computational Method For Aeroelastic Problems In Turbomachines", NASA Contract No. NAS3–26064, April 1993.

Mach number	No motion	Pitching motion 2° midchord	Plunging motion 0.01 normal
0.70	✓	✓	✓
0.80	✓	✓	✓

Table 1.—Simulation matrix. Tenth standard configuration stagger angle = 45°, gap/chord ratio = 1.0, inlet flow angle = 55° (Mach 0.7) and 58° (Mach 0.8) interblade phase angle = 0, reduced frequency = 1.0.

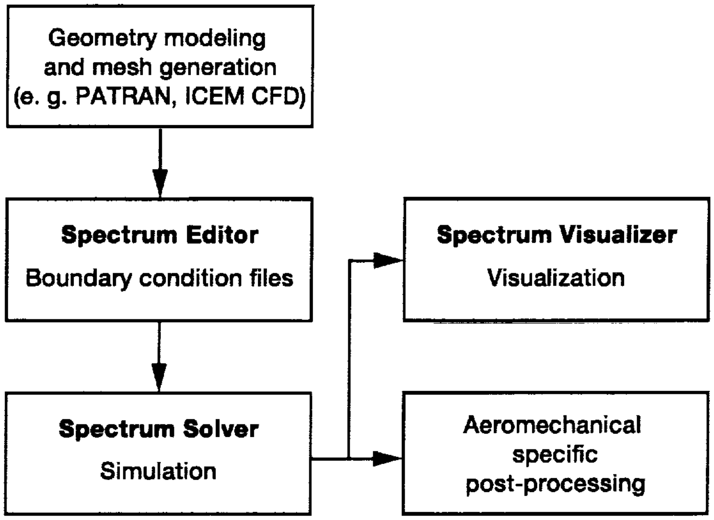

Figure 1.—Aeromechanical simulation process.

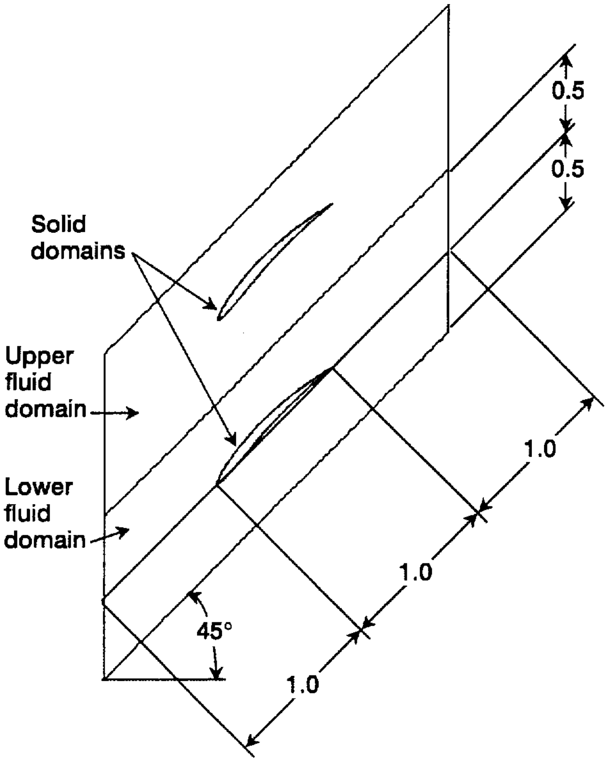

Figure 2.—Fluid and solid domain geometry.

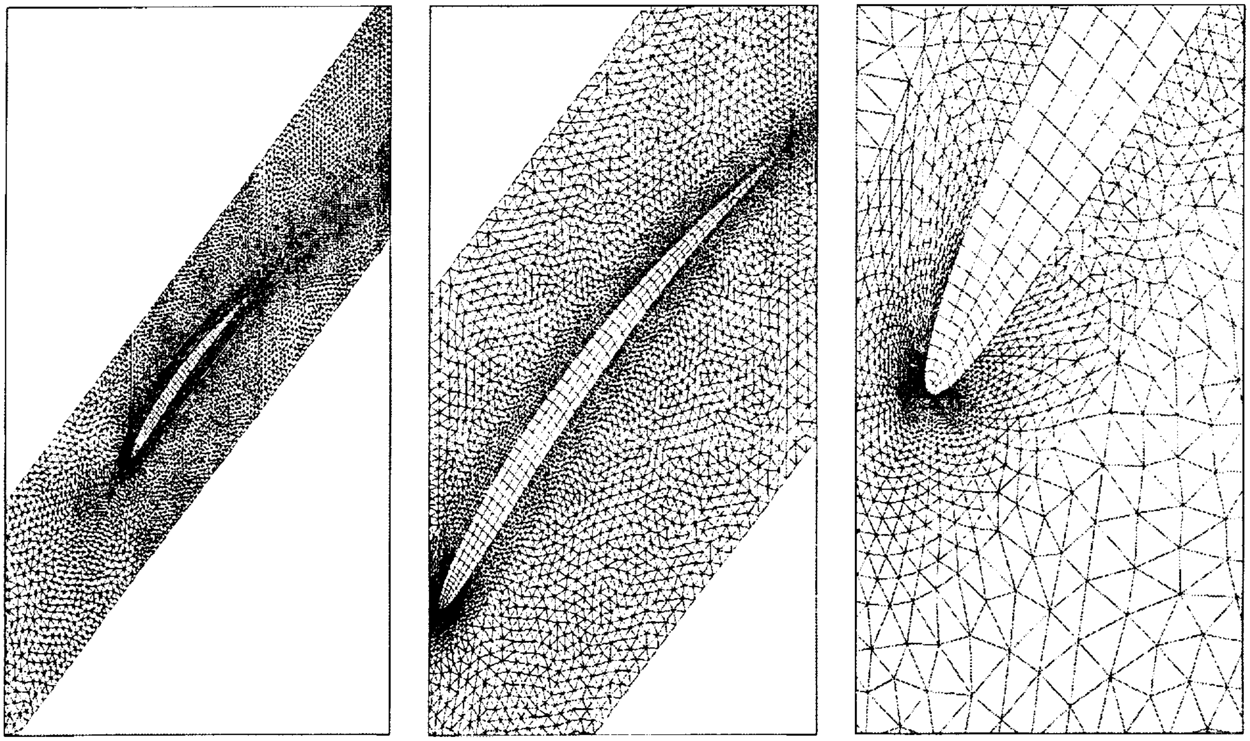

Figure 3.—Fluid and solid domain analysis mesh.

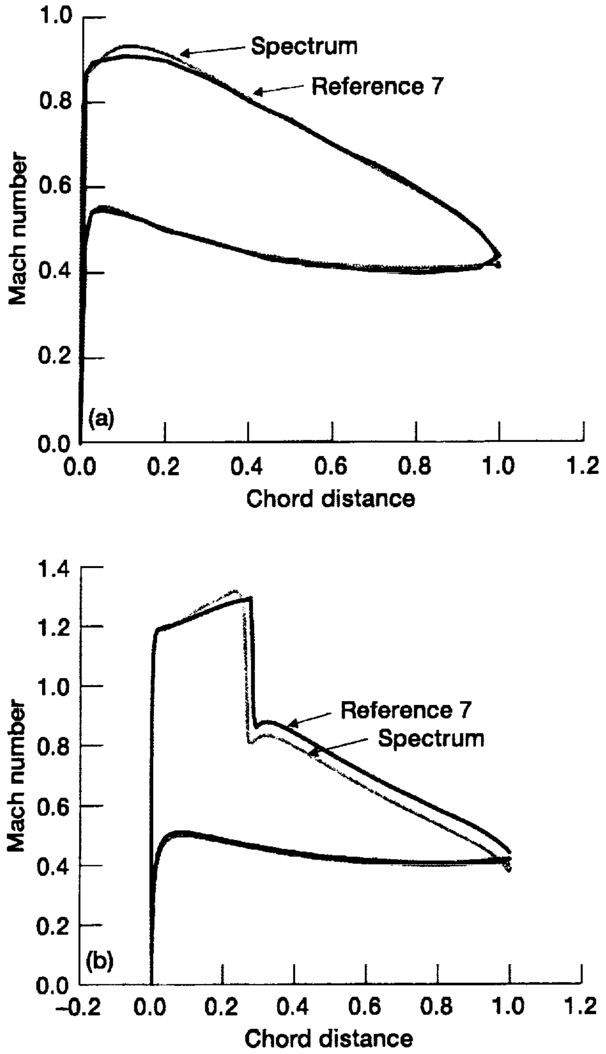

Figure 4.—Airfoil surface steady Mach numbers.
(a) Mach 0.70. (b) Mach 0.80.

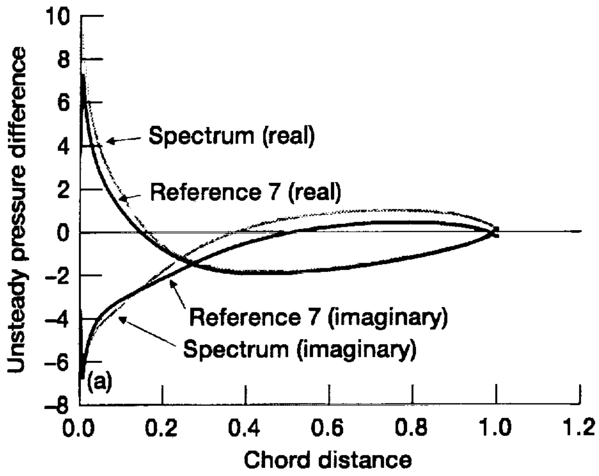

Figure 5.—Mach 0.70, 2 degrees pitching motion.

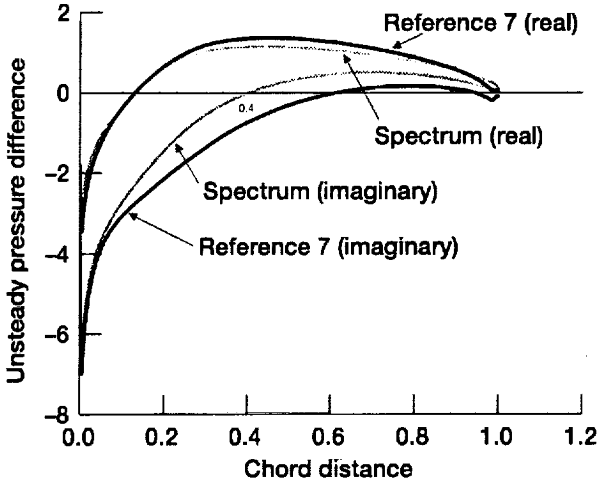

Figure 6.—Mach 0.70, 0.01 plunging motion.

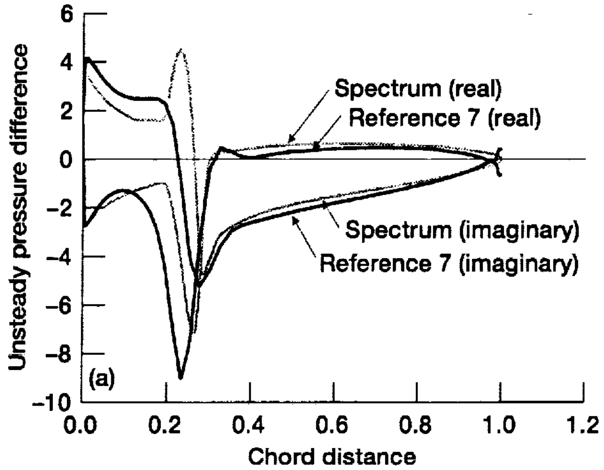

Figure 7.—Mach 0.80, 2 degrees pitching motion.

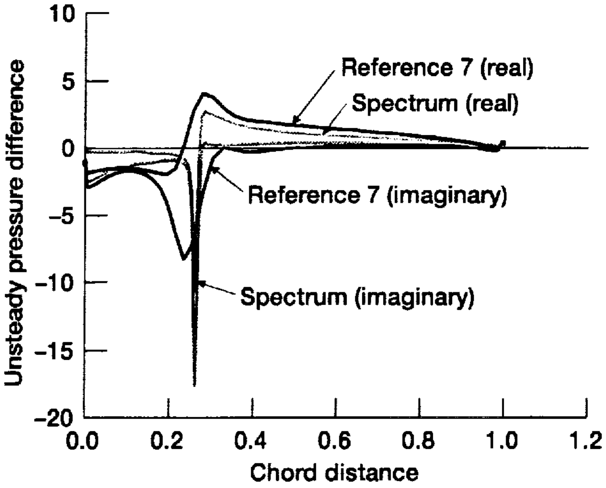

Figure 8.—Mach 0.80, 0.01 plunging motion.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*

2. REPORT DATE
January 2000

3. REPORT TYPE AND DATES COVERED
Technical Memorandum

4. TITLE AND SUBTITLE
Unsteady Cascade Aerodynamic Response Using a Multiphysics Simulation Code

5. FUNDING NUMBERS
WU–509–10–11–00

6. AUTHOR(S)
C. Lawrence, E. Spyropoulos, and T.S.R. Reddy

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

National Aeronautics and Space Administration
John H. Glenn Research Center at Lewis Field
Cleveland, Ohio 44135–3191

8. PERFORMING ORGANIZATION REPORT NUMBER
E–11964

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)

National Aeronautics and Space Administration
Washington, DC 20546–0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
NASA TM—2000-209635

11. SUPPLEMENTARY NOTES

C. Lawrence, NASA Glenn Research Center; E. Spyropoulos, ANSYS, Inc., Sunnyvale, California; and T.S.R. Reddy, Univeristy of Toledo, Toledo, Ohio. Responsible person, C. Lawrence, organization code 5930, (216) 433–6048.

12a. DISTRIBUTION/AVAILABILITY STATEMENT

Unclassified - Unlimited
Subject Category: 01 Distribution: Nonstandard

This publication is available from the NASA Center for AeroSpace Information, (301) 621–0390.

12b. DISTRIBUTION CODE

13. ABSTRACT ***(Maximum 200 words)***

The multiphysics code *Spectrum*™ is applied to calculate the unsteady aerodynamic pressures of oscillating cascade of airfoils representing a blade row of a turbomachinery component. Multiphysics simulation is based on a single computational framework for the modeling of multiple interacting physical phenomena, in the present case being between fluids and structures. Interaction constraints are enforced in a fully coupled manner using the augmented-Lagrangian method. The arbitrary Lagrangian-Eulerian method is utilized to account for deformable fluid domains resulting from blade motions. Unsteady pressures are calculated for a cascade designated as the tenth standard, and undergoing plunging and pitching oscillations. The predicted unsteady pressures are compared with those obtained from an unsteady Euler code referred in the literature. The *Spectrum*™ code predictions showed good correlation for the cases considered.

14. SUBJECT TERMS
Aeropropulsion; Multidisciplinary

15. NUMBER OF PAGES
20

16. PRICE CODE
A03

17. SECURITY CLASSIFICATION OF REPORT	**18. SECURITY CLASSIFICATION OF THIS PAGE**	**19. SECURITY CLASSIFICATION OF ABSTRACT**	**20. LIMITATION OF ABSTRACT**
Unclassified	Unclassified	Unclassified	