

NASA/TM—2002-211351



Overview of High-Fidelity Modeling Activities in the Numerical Propulsion System Simulations (NPSS) Project

Joseph P. Veres
Glenn Research Center, Cleveland, Ohio

June 2002

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—2002-211351



Overview of High-Fidelity Modeling Activities in the Numerical Propulsion System Simulations (NPSS) Project

Joseph P. Veres
Glenn Research Center, Cleveland, Ohio

Prepared for the
Aerospace Numerical Simulation Symposium
sponsored by the National Aerospace Laboratory of Japan
Tokyo, Japan, June 20–22, 2001

National Aeronautics and
Space Administration

Glenn Research Center

June 2002

Acknowledgments

I would like to acknowledge the support of NASA Ames Research Center in providing the resources and the computing time on the NAS computers and the support of Dr. John K. Lytle, head of the Computing and Interdisciplinary Systems Office at NASA Glenn Research Center. I would also like to thank John Adamczyk, Nan-Suey Liu, Mark Turner, and Rob Ryder for their technical contributions to the success of this project.

This report contains preliminary findings, subject to revision as analysis proceeds.

Trade names or manufacturers' names are used in this report for identification only. This usage does not constitute an official endorsement, either expressed or implied, by the National Aeronautics and Space Administration.

Available from

NASA Center for Aerospace Information
7121 Standard Drive
Hanover, MD 21076

National Technical Information Service
5285 Port Royal Road
Springfield, VA 22100

Available electronically at <http://gltrs.grc.nasa.gov/GLTRS>

Overview of High-Fidelity Modeling Activities in the Numerical Propulsion System Simulations (NPSS) Project

Joseph P. Veres
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

Abstract

A high-fidelity simulation of a commercial turbofan engine has been created as part of the Numerical Propulsion System Simulation project. The high-fidelity computer simulation utilizes computer models that were developed at NASA Glenn Research Center, in cooperation with turbofan engine manufacturers. The average-passage (APNASA) Navier-Stokes based viscous flow computer code is used to simulate the 3D flow in the compressors and turbines of the advanced commercial turbofan engine. The 3D National Combustion Code (NCC) is used to simulate the flow and chemistry in the advanced aircraft combustor. The APNASA turbomachinery code and the NCC combustor code exchange boundary conditions at the interface planes at the combustor inlet and exit. This computer simulation technique can evaluate engine performance at steady operating conditions. The 3D flow models provide detailed knowledge of the airflow within the fan and compressor, the high and low pressure turbines and the flow and chemistry within the combustor. The models simulate the performance of the engine at operating conditions that include sea level takeoff and the altitude cruise condition.

Introduction

In the design phase of gas turbine engines, components are designed using computer models that simulate the aerodynamic, thermal and structural characteristics of the component. In the early phases of the design

process, the component characteristics within the thermodynamic cycle system model may not accurately represent the performance of the engine components, and may not model the interactions between the components. The components are later designed, built and tested to verify that the design intent performance specifications have been met. As each component is assembled to form the core engine and ultimately the complete engine system, further verification tests are usually required. Due to component interaction effects such as radial profile variations, the performance of components in an engine system may be different from the performance obtained from isolated component tests. These differences can result in a non-optimum performance match between the engine components. The difference in performance between the isolated component and the performance of the component within the engine system can be the cause of additional iterations in the design and development process. Multiple component and engine build and tests can add \$300 M and 2 years to the design/development cost and time.

NPSS is a system simulation infrastructure designed to provide an integrated computing environment for conducting low and high fidelity engine simulations on a variety of computer platforms (Reference 1). NPSS simulations are characterized by their ability to introduce low and high fidelity component simulation codes (References 2, 3) and combine multi-disciplinary analyses all deployed on heterogeneous computing platforms for fast turnaround.

The goal of the NPSS high fidelity engine simulations is to provide to aircraft engine manufacturers the capability to numerically verify engine performance, including the effects of component interactions, prior to building and testing the engine. This capability will support the development of commercial engines, supersonic business jets, and military programs. NPSS high fidelity simulations can model engine performance at operating conditions throughout the envelope, including sea level takeoff and cruise. High fidelity system simulation can also reduce the amount of expensive engine testing during development, which typically costs in excess of \$15,000 per hour.

Detailed Flow Simulation of a Modern Turbofan Engine

The detailed flow simulation of a modern turbofan engine is one of the key tasks in the NPSS project at NASA Glenn Research Center (NASA GRC). The objective of this task is to perform a 3D flow and chemistry simulation of a modern high-bypass ratio commercial turbofan engine. The goal of the turbofan engine flow simulation task is to perform the high-fidelity computer simulation of the entire engine, at the design point operating condition, in under 24 hours of wall clock time on a parallel computer system.

The GE90 turbofan engine was selected (Fig. 1) and modeled in cooperation with GE Aircraft Engines (References 4, 5, and 6). The detailed flow simulation task leverages from previous efforts at NASA GRC and cooperative efforts with U.S. aerospace industry participants that resulted in the development of the APNASA turbomachinery flow code and the National Combustion Code (NCC). APNASA is NASA's average passage 3-dimensional steady state Navier-Stokes flow code that has been developed to simulate the complex flow within the turbomachinery components of gas turbine engines.

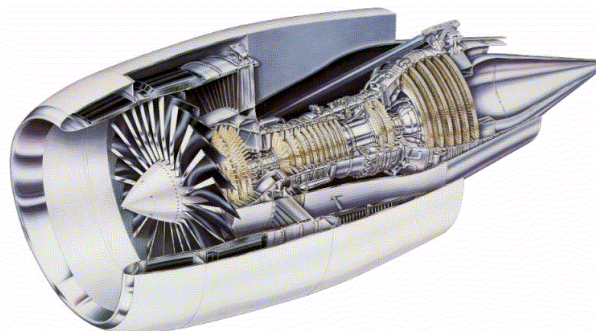


Figure 1. The GE90 high-bypass ratio commercial turbofan engine.

The components within the GE90 turbofan engine's primary flow path were simulated with the 3D Navier-Stokes flow codes. All secondary flows such as compressor bleeds, turbine disk cavity purge and cooling flows have been accounted for to accurately simulate the engine. These secondary flows were modeled macroscopically as source terms, or boundary conditions to the APNASA and NCC codes. The information on these source terms was obtained from detailed descriptions of the bleeds and cooling flow and from low fidelity thermodynamic cycle simulations of the engine system. These source terms provided boundary condition data to the 3D simulations. The radial variations of flow conditions at the exit of the high-pressure compressor have been utilized as the inlet boundary condition for the NCC combustor simulation. The combustor exit conditions, as modeled with the NCC code, were averaged in the circumferential direction and utilized as the inlet boundary conditions for the high-pressure turbine simulation. This method was effective at transferring the combustor exit radial profiles of pressure and temperature to the turbine inlet, and therefore, modeled the steady-state aerodynamic interaction effects between the compressor, combustor and turbine components of the engine.

Compressor Simulation

The fan, booster compressor and the high-pressure compressor (HPC) were successfully modeled with the APNASA flow code. Figure 2 shows the simulation of the fan, booster and HPC with the absolute Mach numbers at mid-stream.

The simulations were run on parallel workstation clusters at NASA Ames Research Center.

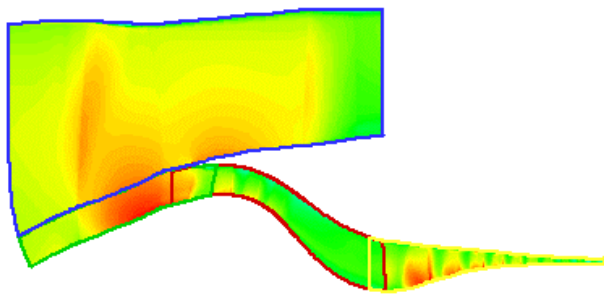


Figure 2. The APNASA flow simulation of the fan, booster and high-pressure compressor.

To simulate the engine turbomachinery in 3D also requires accounting for the multistage aerodynamic blade interaction effects. The APNASA code accounts for most of these effects by calculating the deterministic stresses in the adjacent blade rows and applying these as a body force. Rotor tip clearance and stator leakage flows were also modeled. The inlet total pressure to the APNASA model of the HPC was adjusted to match the exit corrected mass flow in the thermodynamic cycle model. The corrected and physical shaft rotational speeds were held constant at engine cycle values. Figure 3 shows the 21 blade row HPC simulation with the Mach number superimposed onto the blade surfaces. The APNASA simulation of the high-pressure compressor accurately models the performance as measured on compressor test rigs and the engine.

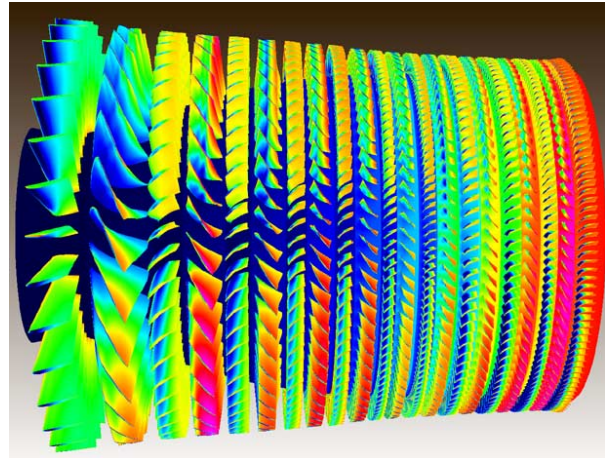


Figure 3. The high-pressure compressor of the high-bypass ratio turbofan engine.

The computer timings were obtained for the converged APNASA simulation of the high-pressure compressor (HPC) on the SGI Origin 3000 computer at Ames Research Center. A total of 504 (400 MHz) processors were used, with 16 to 27 processors per blade row depending on the grid size. The size of the 3D grid is 9.9 million grid nodes. The total wall clock time for 10,000 iterations was 2 hours and 30 minutes.

Combustor Simulation

Modeling of the full combustor was a key part of the turbofan engine simulation project. The aerodynamics and the turbulent combustor flow and chemistry were modeled with the National Combustion Code (NCC). This large-scale calculation also used high performance computers at both NASA GRC and at NASA Ames Research Centers. The NCC is an integrated system of code modules that uses unstructured meshes and can be run on parallel computing platforms for reduced turnaround time. The current version of the NCC consists of several major modules such as baseline flow solver, chemistry, turbulence-chemistry interaction and spray combustion. The code has been used to model the

full combustor from the compressor exit diffuser, to the turbine inlet including all secondary flow regions (Fig. 4).

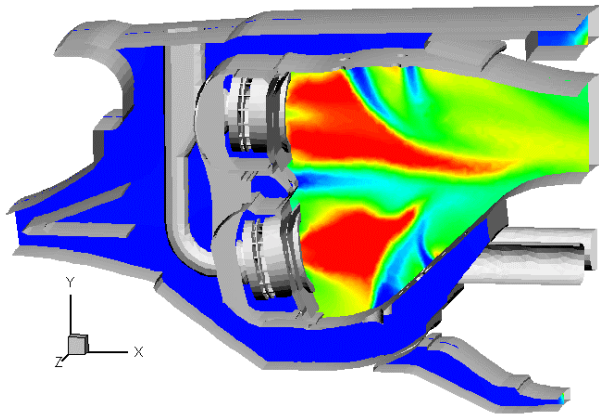


Figure 4. The combustor of the high-bypass ratio turbofan engine.

In a typical turbine engine approximately 80% of the air passes through the combustor and flows through the high-pressure turbine. The remaining 20% is bled off the combustor diffuser for cooling and cavity purge of the turbine blade, vane and disk. The flow path of the complete combustor was modeled with the NCC enabling detailed knowledge of the complex flow field.

The detailed simulation capability provided by the NCC can enable combustor designers to minimize losses through the various regions within the combustor, as well as to optimize cooling and dilution air requirements. Reduced cooling requirements can have a large impact on engine specific fuel consumption. The 3D model of the GE90 combustor is a 24 degree sector representation of the full annulus that is spatially periodic. The NCC combustor model included the compressor exit diffuser, flow swirlers, cooling and dilution holes, inner and outer case and four fuel nozzles. The NCC model consists of a 3D Navier-Stokes flow simulation with heat release. The spray and chemistry models were not utilized in the full engine simulation, but are currently being incorporated into the NCC model. The suc-

cessful NCC simulation increases knowledge about the flow conditions within the combustor and provides the detailed inlet conditions into the cooled high-pressure turbine stage. The NCC simulation enables modeling hot streaks caused by the pattern factor exiting the combustor. However, for this project the total temperature and total enthalpy exiting the combustor were averaged in the circumferential direction to provide the inlet boundary conditions into the subsequent steady state turbine simulation with the APNASA flow code.

Turbine Simulation

The flow simulation through the cooled high and low-pressure turbines and the transition duct (Fig. 5) was successfully accomplished using version 5 of the APNASA flow code. The inlet boundary conditions into the turbine simulation were the radial profiles of total temperature and enthalpy obtained from the exit plane of the combustor. At this plane there is no flow recirculation in the combustor and almost all of the combustion has taken place.

The fully coupled APNASA simulation of the high and low-pressure turbines has provided designers with improved understanding of the detailed flow within the turbine stages.

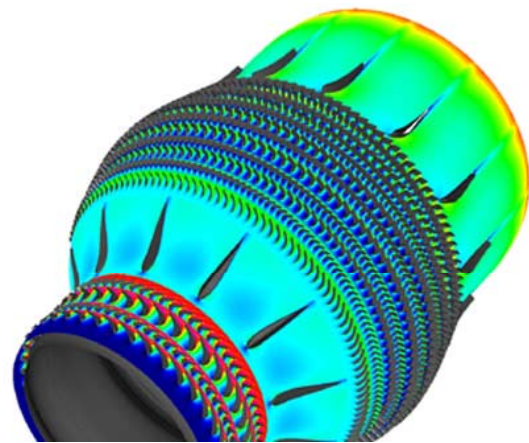


Figure 5. The high and low-pressure turbines of the high-bypass ratio GE90 turbofan engine.

In addition it successfully simulated the interaction between the high and the low-pressure turbines which could not otherwise be modeled in separate simulations (see References 4, 5, and 6 for the detailed description of the turbine simulation). The APNASA combined HP and LP turbine simulation can now be done in 2 hours and 16 minutes on the Chapman computer, which is a 512 node machine with 400 MHz processors at NASA Ames Research Center (References 7, 8, and 9).

Summary

The 3-D flow was successfully simulated in all of the major components of the turbofan engine. The simulation of the complete engine was achieved by utilizing two interface planes to exchange boundary condition data at the combustor inlet and exit. At the two planes of data exchange, the radial profile exit boundary conditions were transferred to the downstream components as inlet boundary conditions. The compressor and turbine were modeled with the APNASA flow code. The combustor was modeled with the National Combustion Code using a gaseous spray model and a Magnussen heat release model.

References

1. "The Numerical Propulsion System Simulation: A Multidisciplinary Design System for Aerospace Vehicles," John K. Lytle, NASA/TM—1999-209194.
2. "Energy Efficient Engine Low Pressure Subsystem Aerodynamic Analysis," Edward J. Hall, Sean R. Lynn, and Joseph P. Veres, Joint Propulsion Conference, 1998, AIAA-98-3119.
3. "Energy Efficient Engine Low Pressure Subsystem Flow Analysis," Edward J. Hall, Sean R. Lynn, Nathan J. Heidegger, and Robert A. Delaney, NASA/CR—1998-206597.
4. "Multistage Simulations of the GE90 Turbine," Mark G. Turner, Paul H. Vitt, David A. Topp, Sohrab Saeidi, Scott D. Hunter, Lyle D. Dailey, and Timothy A. Beach, NASA/CR—1999-209311.
5. "Multistage Simulations of the GE90 Turbine," ASME Paper 99-GT-98, Mark Turner.
6. Turner, Mark G., "Full Analysis of the GE90 Turbofan Primary Flowpath," NASA/CR—2000-209951.
7. "Numerical Propulsion System Simulation (NPSS) 1999 Industry Review," NASA/TM—2000-209795, Lytle, Follen, Naiman, Evans, Veres, Owen, and Lopez.
8. "Numerical Propulsion System Simulation (NPSS) 2000 NPSS Annual Review," NASA/CP—2001-210673, Lytle, Follen, Naiman, Evans, Veres, Owen, and Lopez.
9. "Numerical Propulsion System Simulation (NPSS) 2001 NPSS Annual Review," NASA/TM—2002-211197, Lytle, Follen, Naiman, Evans, Veres, Owen, and Lopez.

REPORT DOCUMENTATION PAGE			<i>Form Approved</i> <i>OMB No. 0704-0188</i>	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (<i>Leave blank</i>)		2. REPORT DATE June 2002	3. REPORT TYPE AND DATES COVERED Technical Memorandum	
4. TITLE AND SUBTITLE Overview of High-Fidelity Modeling Activities in the Numerical Propulsion System Simulations (NPSS) Project			5. FUNDING NUMBERS WU-704-31-63-00	
6. AUTHOR(S) Joseph P. Veres				
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-13169	
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-2002-211351	
11. SUPPLEMENTARY NOTES Prepared for the Aerospace Numerical Simulation Symposium sponsored by the National Aerospace Laboratory of Japan, Tokyo, Japan, June 20-22, 2001. Responsible person, Joseph P. Veres, organization code 2900, 216-433-2436.				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified - Unlimited Subject Categories: 07 and 64 Available electronically at http://gltrs.grc.nasa.gov/GLTRS This publication is available from the NASA Center for AeroSpace Information, 301-621-0390.			12b. DISTRIBUTION CODE	
13. ABSTRACT (<i>Maximum 200 words</i>) A high-fidelity simulation of a commercial turbofan engine has been created as part of the Numerical Propulsion System Simulation Project. The high-fidelity computer simulation utilizes computer models that were developed at NASA Glenn Research Center in cooperation with turbofan engine manufacturers. The average-passage (APNASA) Navier-Stokes based viscous flow computer code is used to simulate the 3D flow in the compressors and turbines of the advanced commercial turbofan engine. The 3D National Combustion Code (NCC) is used to simulate the flow and chemistry in the advanced aircraft combustor. The APNASA turbomachinery code and the NCC combustor code exchange boundary conditions at the interface planes at the combustor inlet and exit. This computer simulation technique can evaluate engine performance at steady operating conditions. The 3D flow models provide detailed knowledge of the airflow within the fan and compressor, the high and low pressure turbines, and the flow and chemistry within the combustor. The models simulate the performance of the engine at operating conditions that include sea level takeoff and the altitude cruise condition.				
14. SUBJECT TERMS Gas turbine engines; Computational fluid dynamics; Turbocompressions; Turbines; Combustion chambers			15. NUMBER OF PAGES 11	
			16. PRICE CODE	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT	