

NASA/CR—2007-214687



Preliminary Assessment of Turbomachinery Codes

Quamrul H. Mazumder, Ph.D.
University of Michigan Flint, Flint, Michigan

March 2007

NASA STI Program . . . 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 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 NASA Aeronautics and Space Database 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.
- **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 also include creating custom thesauri, building customized databases, organizing and publishing research results.

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

- Access the NASA STI program home page at <http://www.sti.nasa.gov>
- E-mail your question via the Internet to help@sti.nasa.gov
- Fax your question to the NASA STI Help Desk at 301-621-0134
- Telephone the NASA STI Help Desk at 301-621-0390
- Write to:
NASA Center for AeroSpace Information (CASI)
7115 Standard Drive
Hanover, MD 21076-1320

NASA/CR—2007-214687



Preliminary Assessment of Turbomachinery Codes

Quamrul H. Mazumder, Ph.D.
University of Michigan Flint, Flint, Michigan

Prepared under Cooperative Agreement NNC06ZA42A

National Aeronautics and
Space Administration

Glenn Research Center
Cleveland, Ohio 44135

March 2007

Acknowledgments

The author would like to thank Joseph Veres and Dhanireddy Reddy for their outstanding support and cooperation during the fellowship at NASA Glenn Research Center. They have provided all the required resources, guidance, and valuable input during this study. The CFD code developers and users were also helpful in providing insight about their codes. I would like to thank Aamir Shabir, Chunil Hah, Jim Heidmann, Bob Boyle, Wei-Ming To, and Rod Chima for their support. This work was supported by the Aerothermodynamics Subsonic Fixed Wing element of the Fundamental Aeronautics program of NASA.

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

This work was sponsored by the Fundamental Aeronautics Program
at the NASA Glenn Research Center.

Level of Review: This material has been technically reviewed by NASA expert reviewer(s).

Available from

NASA Center for Aerospace Information
7115 Standard Drive
Hanover, MD 21076-1320

National Technical Information Service
5285 Port Royal Road
Springfield, VA 22161

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

Preliminary Assessment of Turbomachinery Codes

Quamrul H. Mazumder, Ph.D.
University of Michigan Flint
Flint, Michigan 48502

Introduction

Flow inside multistage turbomachinery is a complex phenomenon due to unsteady flow phenomenon and complex passage configurations. Accurate prediction of the flow behavior requires careful consideration of the parameters that affects the flow field. Computational fluid dynamics (CFD) codes have been used by aerodynamicists to predict the flow behavior that requires significantly large computational memory space. With the advancement of computers, advanced CFD codes have been developed to analyze complex three-dimensional flow behavior in turbomachinery. At NASA Glenn Research Center, Cleveland, Ohio, a number of CFD codes have been developed and are being used to predict flow behavior inside compressors, turbines and fans. These state-of-the-art CFD codes use thermodynamic properties of fluid, Navier-Stokes equation and turbulence models to simulate flow behavior inside turbomachinery.

Objective

The objective of this report is to assess different CFD codes developed and currently being used at Glenn Research Center to predict turbomachinery fluid flow and heat transfer behavior. This report will consider the following codes: APNASA, TURBO, GlennHT, H3D, and SWIFT. Each code will be described separately in the following section with their current modeling capabilities, level of validation, pre/post processing, and future development and validation requirements. This report addresses only previously published applications and validations of the codes. However, the codes have been further developed to extend the capabilities of the codes.

APNASA

APNASA is a three-dimensional, steady-state, time-averaged Navier-Stoke code for multistage compressor analysis developed in 1985 at NASA Glenn Research Center by John Adamczyk. The averaging process is based on a sequence of mathematical filtering or averaging of Navier-Stokes equations, continuity equation, energy equation and equation of state resulting a system of equation for the governing flow behavior. The filtering operator used in the construction of the Reynolds averaged form of Navier-Stokes equation ensemble-averages consecutive samples of data taken over one shaft revolution (ref. 1).

Three different averaging operators are applied to solve the Navier-Stokes equation. The first operator referred to as “ensemble averaging” used to eliminate the need to resolve, in details the structure of the turbulent flow that yields the Reynolds-averaged form of Navier-Stokes equation (ref. 2). The second operator is the time averaging operator that averages flow fields with respect to stator frame of reference and with respect to rotor frame of reference were used to define the flow fields in inside a multistage turbomachine. The third operator averages the out the passage to passage variations in the flow field considering the global effect of this variation on the flow field.

The capabilities of this code includes prediction of interaction between stationary and rotating components of multistage turbomachinery (i.e., fan, compressor, turbine, etc.). The computational fluid dynamics code is a steady state; viscous solver for Navier-Stokes equation uses two equation turbulence model (k - ϵ). Variable thermodynamic and physical properties (c_p , c_v , γ) of fluids at different temperature are used during analysis.

As the code is written in FORTRAN 90, it can run in any computer with sufficient memory for the problem. APNASA is a stand alone CFD code with its own built-in grid generator, pre and post processors that eliminates the need for additional software to analyze a problem.

The uniqueness of APNASA lies in its ability to model the aerodynamic effect of neighboring blade rows on a given blade row. It has been successfully used to simulate a number of geometries from commercial aero-engine companies (e.g., ref. 3) as well as NASA's own research compressors (e.g., ref. 4). The code also allows modeling the flow injection or bleeding on blade surfaces and ending walls via surface boundary conditions (e.g., ref. 5). These bleed ports sometimes can significantly impact the aerodynamic performance of turbomachinery. APNASA has its own mesh generator which generates meshes consistent with the requirements of its average passage flow model. It also has its own post processor which generates aerodynamic performance on both one-dimensional and axisymmetric basis.

GlennHT

Glenn HT (ref. 6) is a multi-block steady-state Navier-Stokes fluid flow and heat transfer code that can predict the fluid flow and heat transfer characteristics in turbomachinery. The code has been used primarily for heat transfer applications such as internal cooling flow, film cooling flow, turbine blade tip heat transfer, and flow characteristics in blade tip regions.

The overall methodology is based on three design choices: (1) Globally unstructured—locally structured multiblock grid system (2) finite volume discretization of the governing equations of fluid flow, and (3) an explicit, multistage time stepping scheme combined with multigrid convergence acceleration for marching solutions to steady state. GlennHT was derived from a computer code TRAF3D that has demonstrated effective finite volume discretization and multigrid convergence acceleration of flows in turbomachinery.

The original version of GlennHT and TRAF3D used Baldwin-Lomax turbulence model that was later replaced by $k-\omega$ turbulence model in 1994. As the Baldwin-Lomax model requires length scale (distance from the wall), the model capabilities were limited in prediction of near-wall flow regions. The introduction of $k-\omega$ model eliminated the limitations of previous Baldwin-Lomax model and demonstrated computational efficiency and robustness in several turbomachinery applications.

A separate module is used to handle the "management" of the blocks and communication between blocks in GlennHT. All operations on data in the block are performed by in separate subroutines that do not have any information about the neighboring blocks. The modularity of the multiblock grid system makes it easier to modify or experiment with the discretization and relaxation scheme.

The unique capability of GlennHT is to analyze heat transfer characteristics of turbomachinery although the code is also capable of analyzing aerodynamics flow behavior in flow passages. The code was applied to a wide variety of turbine convective heat transfer problems including tip clearance flows, internal cooling passage flows, external turbine blade flows including film cooling. Examples of a few validations are described in the following paragraph.

Flow and heat transfer analysis in rectangular holes with ribs and bleed holes using GlennHT was performed and compared with experimental data (ref. 7). The computational results of GlennHT showed reasonably good agreement with the experimental results that can be used in prediction of heat transfer in coolant passages. Blade tip heat transfer of a first stage large power generation turbine blade was analyzed in reference 8 and compared with both sharp edged and radiused edge blades. The computational results of blade tip heat transfer showed good agreement with the radiused edge blade than the sharp edged blade. Flow over the sharp edged blade tip showed separation and reattachment due to vorticity that was eliminated by using radiused edge.

The code was recently developed to improve predictions in a broader range of flow problems. Conjugate heat transfer capability has been incorporated using boundary element method to allow simultaneous computation of fluid and solid heat transfer without requiring a solid volume grid (ref. 9). This capability was extended to layer solids to analyze turbine blades with thermal barrier coating and solids with variable thermal conductivity. Some of the other development efforts currently being

undertaken include incorporation of Reynolds Stress model, automatic topology generation, incorporation of unsteady flow capability.

H3D

A pressure-based solver for incompressible and compressible three-dimensional Navier-Stokes equation for turbomachinery. This CFD code uses two-equation k - ϵ turbulence model, RANS, unsteady Reynolds Averaged Navier-Stokes, and Large Eddy Simulation models.

For the Large eddy simulation, a standard dynamic model is used for the sub grid stress tensor (ref. 10).

A study was conducted to compare the numerical output of H3D program where the governing equations were solved in non orthogonal curvilinear coordinate system by applying higher order discretization schemes for the convection terms to reduce the numerical diffusion (ref. 11). An algebraic Reynolds stress model modified for the stream line curvature was used in the study. Comparison of the analysis with experimental data showed satisfactory agreement in predicting: three-dimensional viscous flow behavior inside turbine blade passages, boundary layer separation and attachment locations, secondary flow characteristics, and local pressure loss.

To evaluate the cavitation inception, a numerical study was conducted to investigate the flow field near blade tip of a ducted propeller (ref. 12). Due to Unsteady flow behavior near the blade tip section, a large eddy simulation was applied. A local low pressure core was observed when the tip leakage vortex is stretched and twisted as the tip leakage vortex from the adjacent blade interacts with the shed vortices in the rotor wake region. Comparison of the LES analysis with experimental data shows good qualitative agreement.

To improve the understanding of the flow mechanism that leads to the onset of short length scale rotating stall in a transonic compressor, a numerical and experimental investigation was conducted (ref. 13). As the rotor operates near stall, the flow field becomes unsteady due to oscillation of tip clearance vortices and their interactions with the shocks in the passages. The measured data agreed with the H3D analysis that the unsteady behavior due to tip clearance vortex oscillation is much larger than those of purge shock only.

The code has been used to develop concept of swept rotor for high speed fan/compressor stages jointly with the General Electric Aviation and the U.S. Air Force. The code has been extensively applied to study flow and cavitation characteristics in pump stages in the current Space Shuttle Main Engine and future space engine development programs.

The code has been successfully applied to develop a very high-pressure-ratio centrifugal compressor stage. Calculated unsteady flow field in a single stage centrifugal compressor agrees well with the experimental data obtained at the DLR, Germany.

TURBO

An implicit finite volume code uses Reynolds-Averaged Navier-Stokes equation solver for turbomachinery. The CFD code uses multiblock structured grid with arbitrary block connectivity. It focuses on unsteady flow behavior caused by the relative motion between blade rows in a multi-stage turbomachines. A two equation (k - ϵ) turbulence model in rotating frame is used in the code. Convective fluxes are evaluated by the high resolution Roe scheme and a sliding interface is applied to capture the time-accurate relative motion between blade rows (Wang). A real gas model is used to account for temperature variation where the specific heat coefficient is a function of temperature.

The capabilities of TURBO includes unsteady multiple blade row calculation, flutter simulation, hybrid grid topology, real gas and perfect gas models used in the analysis.

A preprocessor, GUMBO (Graphical Unstructured MultiBlock Omnitool) is used for blocking, block connectivity, and boundary condition information input to TURBO.

TURBO was used to study the interaction effects of reflective waves between blade rows in a transonic turbine stage (ref. 14). Wake blade analysis and coupled stage analysis showed that wake blade analysis captures the incident waves into a blade row but does not capture the effect of waves that are reflected between blade rows. Whereas, the stage analysis captured the relevant physical interactions including reflective waves. The analysis results showed good agreement with the experimental data demonstrating incident waves to be the dominant contributor to unsteadiness on the blade. TURBO was also used to demonstrate the feasibility of stall inception and flow control computations of the NASA Stage 35 compressor with a full annulus.

Experimental validation of predicted flow in a transonic compressor with an IGV row was performed in reference 15 using TURBO. The Euler grid refinement studies showed over prediction of IGV steady surface pressures while underpredicting the magnitude of the unsteady component, primarily the trailing region of the grid. Although the rotor-IGV simulation exhibited good qualitative agreement, there were quantitative differences in viscous prediction due to grid limitations. Grid refinement is expected to improve the predictions by TURBO to match experimental data.

SWIFT

A multiblock, steady state, three-dimensional CFD code for analysis of flows in turbomachinery. Using explicit finite difference techniques, this code solves thin layer Navier Stokes equation. SWIFT is a multiblock three-dimensional version of RVC3D which is a quasi- three-dimensional Navier Stokes code for analysis of blade to blade flows in turbomachinery. The multiblock capabilities include C-grid around the blades and H-grid in upstream, O-grid in hub or tip clearance region and mixing plane between rows. It can be used for linear cascades or annular blade rows with or without rotation. Typical applications includes linear cascades, axial compressors and turbines, isolated blade rows, centrifugal and mixed flow impellers, radial diffusers, pumps and ducts.

SWIFT uses Cartesian coordinate system with rotations about x-, y-, and z-axes to solve the Navier Stokes equation. Using the thin layer assumption, stress and viscous terms are neglected but all cross channel viscous terms are retained. SWIFT uses central differencing scheme with artificial viscosity, H-CUSP upwind method or AUSM and upwind scheme. The turbulence models used are Baldwin-Lomax (algebraic), Cebeci-Smith (algebraic), Wilcox's $k-\omega$ (two-equation) or Wilcox's $k-\omega$ plus Mentor SST models.

Using variable time step and implicit residual smoothing accelerated convergence is attained. The pre-processor used for grid generation are TCGRID and are stored in PLOT3D format. The solutions are in text format that can be graphically displayed using PLOT3D. The code is written in FORTRAN 90 and therefore, a FORTRAN 90 compiler is required to run the program.

SWIFT has been used for analysis and design of fan, turbine blades and turbine end wall heat transfer (refs. 16 to 19).

TRVC3D and TRVCQ3D

These codes are specialized versions of RVC3D and RVCQ3D that were modified to account for factors typically not included in turbulence models. The modifications are important when laminar flow is present. In addition to transition start and length models, relaminarization, and freestream turbulence effects on laminar flow are modeled. The effects of surface roughness on both heat transfer and losses have been investigated. The work has been primarily focused on heat transfer, but blade row losses have also been investigated.

Both TRVC3D and TRVCQ3D used C-grids exclusively. For improved orthogonality, non-matching grid spacing is used along the C-grid cut line. The quasi- three-dimensional version, TRVCQ3D, has algebraic and the $k-\omega$ two equation turbulence models. The fully three-dimensional code, TRVC3D, has only algebraic turbulence models, (Baldwin-Lomax and Cebeci-Smith). There is no grid in the clearance

region, and only a primitive clearance model is used. The boundary condition for the C-grid in the clearance region equates suction and pressure side static pressures to allow flows in the clearance region.

Results from the three-dimensional code for rotor geometry with tip clearance are given in references 20 to 22 show comparisons with data for two-dimensional calculations for smooth and rough surfaces.

Summary and Recommendation

A number of different CFD (computational fluid dynamics) codes have been developed and used at NASA Glenn Research Center to evaluate turbomachinery aerodynamics and thermal performance (table 1). These codes have unique capabilities and were developed by one or a group of individual researchers at Glenn Research Center. Several of these codes are research codes having sparse documentation, making it challenging for other potential users to understand the code methodology and/or run the code for flow analysis. The usage of these research codes can be drastically improved with proper documentation. One recommendation to improve the current situation is for each code developer to develop a comprehensive user manual and train new users about the code to a level that the new users are capable of performing analyses with these codes. Potential future work is to evaluate the capabilities of commercial CFD codes and to compare them to NASA codes for simulation accuracy of turbomachinery aerodynamics and heat transfer.

TABLE 1.—SUMMARY OF TURBOMACHINERY CODE ASSESSMENT

	APNASA	GlennHT	H3D	TURBO	SWIFT
Code methodology	Compressible, steady state, Navier-Stokes code (RANS)	Compressible unsteady Navier-Stokes, Multi-stage Runge-Kutta (explicit), Multigrid method, Finite volume, Central differencing, Artificial dissipation, 2nd order accuracy	Pressure-based, steady and unsteady Navier-Stokes solver over all speed regime.	Implicit finite volume time marching, high order Roe's scheme, Newton's method of solution and Gauss-Seidel iteration for matrix inversion.	Explicit Runge-Kutta Central differences or AUSM+upwind
Level of fidelity	Steady, viscous, three-dimensional, multistage	High fidelity three-dimensional solutions	Steady, unsteady RANS, LES, isolated or multi stage turbomachinery	High fidelity three-dimensional unsteady Reynolds Averaged Navier-Stokes solver, second order in time and third order in space.	High
Turbulent model	Two equation $k-\epsilon$ model	$k-\omega$ two-equation model	Two-equation or LES with dynamic subgrid stress model.	$k-\epsilon$ (CMOTT extension) and low Re $k-\epsilon$ models.	Baldwin-Lomax Cebeci-Smith $k-\omega$
Pre-processor	MMESH, APG	GridPro, GUI	Own grid generator	Mesh generation code, GUMBO, and interpolation code.	TCGRRID grid generator
Post-processor	Built-in own post processor	GUI, Tec plot, Field view	Fast	GUMBO, Visual 3, and application-specific user-developed specialized code.	Any commercial CFD visualization package

TABLE 1.—Concluded.

Unique features	Simulates a blade passage embedded in a multistage environment	General multiblock capability, Near-Wall modeling for heat transfer	Incompressible and compressible flows. RANS and LES.	Multi-block, parallel, 3D, unsteady Navier-Stokes solver, real gas model, tip clearance model.	Fast, easy to use, handles linear cascades, gridded tip clearances, multistage calculations distributed to over 200 users.
Capabilities	Steady, aerodynamic	Convective heat transfer and aerodynamics for general geometries, conjugate heat transfer, unsteady flows	Steady and unsteady. Isolated blade row or multiple blade rows.	Capable of solving multi-stage axial turbo machineries, blade fluttering in fans, and flow injection flow control of compressors.	axial and radial turbines, fans, compressors, and pumps
Validations performed in turbo machinery (Show a list of validations)	Hub leakage effect on Rotor 37, Low speed axial compressor, GE90, several other commercial and military geometries	Tip clearance flows Internal coolant passage flows Film-cooled turbine flows Turbine aerodynamics	Compressor, turbines, centrifugal machines, pumps, submarine propellers, compressor and turbine stages, pump stages.	NASA Rotor 35, Stage 35, and UEET 2.5 Stage POC compressor	Rotors 33, 35,37, 67 Stage 35 SSME turbine Goldman cascade DLR turbine Low speed centrifugal. Used to design Supersonic Through flow Fan and Trailing Edge Blowing Fan
Future development needs	Multiblock capability to handle complicated geometries (e.g. centrifugal flow path with splits, etc.)	Multi-blade-row turbine unsteady flow capability General interface capability	Further refinement and validation of LES and DNS for turbomachinery flows.	Improve turbulence model, heat transfer capability, parallel efficiency, and needs code integration with WIND.	Splitters axisymmetric capability for far field

References

1. Van Zante, D.Z., Strazisar, A.J., Wood, J.R., Hathaway, M.D., Okiishi, T.H., “Recommendations for Achieving Accurate Numerical Simulation of Tip Clearance Flows in Transonic Compressor Rotors,” ASME paper 99–GT–390.
2. Hathaway, M.D., “Self-Recirculating Casing Treatment Concept for Enhanced Compressor Performance,” NASA/TM—2002-211569.
3. Turner, M.G., Reed, J.A., Ryder, R., Veres, J.P., “Multi-Fidelity Simulation of a Turbofan Engine With Results Zoomed Into Mini-Maps for a Zero-D Cycle Simulation,” Paper no. GT2004–53956. ASME Turbo Expo 2004, June 14–17, 2004, Vienna, Austria.
4. Adamczyk, J.J., “Aerodynamic Analysis of Multistage Turbomachinery Flows in Support of Aerodynamic Design,” Journal of Turbomachinery, vol. 122, no. 2, pp. 189–217, April 2000.
5. Adamczyk, J.J., “Modern Equation for Simulating Flow in Multistage Turbomachinery,” Paper no. 85–GT–226, ASME Gas Turbine Conference, March 18–25, 1985, Houston, Texas.
6. GlennHT Developer Manual. (unpublished)

7. Rigby, David L., Steinhilber, E., Ameri, Ali A., “Numerical Prediction of Heat Transfer in Channel with Ribs and Bleed.”
8. Ameri, A.A., Bunker, R.S., “Heat Transfer and Flow on the First-Stage Blade Tip of a Power Generation Gas Turbine: Part 2- Simulation,” ASME Journal of Turbomachinery, vol. 122, April 200, pp. 272–277.
9. Heidmann, J.D., Ameri, A.A., Garg, V.K., Rigby, D.L., “Improved Modeling Capabilities in Glenn-HT—The NASA Glenn Research Center General Multiblock Navier-Stokes Heat Transfer Code. Presented at JANNAF CS/APS/PSHS/MSS Joint Meeting, Colorado Spring, CO, Dec. 1–5, 2003.
10. Germano, M., Piomelli, U., Moin, P., and Cabot, W.H., “A Dynamic Subgrid-Scale Eddy-Viscosity Model,” Journal of Fluid Mechanics, vol. A3, pp. 170–176, 1991.
11. Hah, C. “A Navier-Stokes Analysis of Three-Dimensional Turbulent Flow Inside Turbine Blade Rows at Design and Off-Design Conditions” Journal of Engineering for Gas Turbines and Power, vol. 106, April 1984, pp. 421–429.
12. Hah, C., and Lee, Yu-Tai, “Unsteady Tip Leakage Vortex Phenomena in A Ducted Impeller.”
13. Hah, C., Bergner, J., Schiffer, Heinz-Peter, “Short Length Scale Rotating Stall Inception in a Transonic Axial Compressor—Criteria and Mechanism,” Paper no. GT2006–90045. ASME Turbo Expo 2006, May 8–11, 2006, Barcelona, Spain.
14. Bartner, J.W., Vitt, P.H., Chen, J.P., “Interaction Effects in A Transonic Turbine Stage,” Paper no. 200–GT–0376, ASME Turbo expo 2000, May 8–11, 2000, Munich, Germany.
15. Richmann, M. and Fleeter S., “Navier- Stokes Simulation of IGV-Rotor-Stator Interactions in a Transonic Compressor,” AIAA–2000–3379, 2000.
16. Chima. R.V., “A Three-Dimensional Unsteady CFD Model of Compressor Stability,” ASME Paper GT2006–90040, May 2006. Also NASA/TM—2006-214117.
17. Chima, R.V. and Liou, M.-S., “Comparison of the AUSM+ and H-Cusp Schemes for Turbomachinery Applications,” AIAA Paper AIAA–2003–4120, June 2003. Also NASA/TM—2003-212457.
18. Chima, R.V., “Calculation of Multistage Turbomachinery Using Steady Characteristic Boundary Conditions,” AIAA Paper 98–0968, Jan. 1998. Also NASA TM-1998-206613.
19. Chima, R.V., “Viscous Three-Dimensional Calculations of Transonic Fan Performance in CFD Techniques for Propulsion Applications, AGARD Conference Proceeding No. CP–510, AGARD, Neuilly-Sur-Seine, France, Feb. 1992, pp. 21–1 to 21–9, Also NASA TM-103800.
20. Boyle, R.J., and Giel, P.W., “Three-Dimensional Navier-Stokes Heat Transfer Predictions for Turbine Blade Rows,” AIAA Journal of Propulsion and Power vol. 11, no. 6, pp. 1179–1186.
21. Boyle, R.J., and Simon, F.F., “Mach Number Effects on Turbine Blade Transition Length Prediction,” ASME Journal of Turbomachinery vol. 121, no. 4, pp. 694–702.
22. Boyle, R.J., and Senyitko, R.G., “Measurements and Predictions of Surface Roughness Effects on Turbine Vane Aerodynamics,” ASME paper GT–2003–38580.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

The 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 Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number.

PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.

1. REPORT DATE (DD-MM-YYYY) 29-03-2007		2. REPORT TYPE Final Contractor Report		3. DATES COVERED (From - To)	
4. TITLE AND SUBTITLE Preliminary Assessment of Turbomachinery Codes				5a. CONTRACT NUMBER	
				5b. GRANT NUMBER	
				5c. PROGRAM ELEMENT NUMBER	
6. AUTHOR(S) Mazumder, Quamrul, H., Ph.D.				5d. PROJECT NUMBER NNC06ZA42A	
				5e. TASK NUMBER	
				5f. WORK UNIT NUMBER WBS 561581.02.08.03.21.03	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) University of Michigan Flint 303 E. Kearsley St. Flint, Michigan 48502				8. PERFORMING ORGANIZATION REPORT NUMBER E-15871	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001				10. SPONSORING/MONITORS ACRONYM(S) NASA	
				11. SPONSORING/MONITORING REPORT NUMBER NASA/CR-2007-214687	
12. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified-Unlimited Subject Categories: 02 and 07 Available electronically at http://gltrs.grc.nasa.gov This publication is available from the NASA Center for AeroSpace Information, 301-621-0390					
13. SUPPLEMENTARY NOTES					
14. ABSTRACT This report assesses different CFD codes developed and currently being used at Glenn Research Center to predict turbomachinery fluid flow and heat transfer behavior. This report will consider the following codes: APNASA, TURBO, GlennHT, H3D, and SWIFT. Each code will be described separately in the following section with their current modeling capabilities, level of validation, pre/post processing, and future development and validation requirements. This report addresses only previously published and validations of the codes. However, the codes have been further developed to extend the capabilities of the codes.					
15. SUBJECT TERMS Compressors; Turbines; Computational fluid dynamics; Simulation; Flow distribution					
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES 13	19a. NAME OF RESPONSIBLE PERSON Joseph Veres
a. REPORT U	b. ABSTRACT U	c. THIS PAGE U			19b. TELEPHONE NUMBER (include area code) 216-433-2436

