

**Glenn-HT: The NASA Glenn Research Center General Multi-Block  
Navier-Stokes Heat Transfer Code**

**R. E. Gaugler  
Chief, Turbine Branch  
NASA Glenn Research Center  
Cleveland, OH**

**ABSTRACT**

For the last several years, Glenn-HT, a three-dimensional (3D) Computational Fluid Dynamics (CFD) computer code for the analysis of gas turbine flow and convective heat transfer has been evolving at the NASA Glenn Research Center. The code is unique in the ability to give a highly detailed representation of the flow field very close to solid surfaces in order to get accurate representation of fluid heat transfer and viscous shear stresses. The code has been validated and used extensively for both internal cooling passage flow and for hot gas path flows, including detailed film cooling calculations and complex tip clearance gap flow and heat transfer. In its current form, this code has a multiblock grid capability and has been validated for a number of turbine configurations. The code has been developed and used primarily as a research tool, but it can be useful for detailed design analysis. In this paper, the code is described and examples of its validation and use for complex flow calculations are presented, emphasizing the applicability to turbomachinery for space launch vehicle propulsion systems.

**INTRODUCTION**

The flow environment in the turbine section of a gas turbine engine is extremely harsh and complex, Figure 1. The hot gas flow entering the turbine from the combustor is highly turbulent, and in most cases, at a temperature above the melting point of the metallic materials in the turbine. The very survival of the engine requires that the turbine be adequately cooled over the entire range of system operation. However, system weight and efficiency requirements dictate that the cooling configuration selected have a minimal impact on those system parameters. Therefore, the turbine designer must have an accurate knowledge of the detailed distribution of critical flow parameters such as pressure, temperature, and heat transfer coefficient, both in the hot gas stream and in the turbine cooling flow path. A lot of this knowledge is derived from the company experience base, but more and more, the fine details come from a computational fluid dynamics (CFD) analysis of the design. A turbine engine for a space launch vehicle propulsion system poses a particular problem for the designer in that the vehicle is continually accelerating, requiring that the turbomachinery operate effectively over a broad range of flow conditions. This adds to the attractiveness of accurate CFD tools to extrapolate beyond the current experience base.

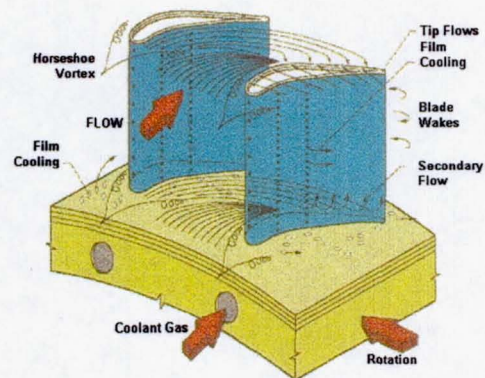


Fig. 1 Typical Turbine Flow Problems

As CFD researchers developed the capability to analyze the complex flow fields in gas turbine engine rotating components, the emphasis was always on modeling the aerodynamics correctly, providing good representation of the pressure distribution and flow patterns in the machine. However, the capability to accurately model near-wall viscous phenomena such as convective heat transfer and wall shear stress was lacking, primarily because of the fine resolution of computational grids needed to accomplish this. Near-wall phenomena require accurate computations of the gradients of physical



variables near the wall, and this requires that the grid near the wall be more closely spaced and more nearly orthogonal than required for mainstream variables.

Since the mid 1970's, the NASA Lewis Research Center (LeRC), now the NASA Glenn Research Center (GRC), has had a very strong CFD research and development effort in place, but it was not until the late 1980's that researchers finally addressed the near-wall issues. The seminal publication in this area was presented by Robert J. Boyle, of NASA LeRC, at the 35<sup>th</sup> ASME International Gas Turbine and Aeroengine Congress and Exposition, Brussels, Belgium, June 11-14, 1990<sup>(1)</sup>. Boyle used the quasi 3D viscous flow code developed by Chima<sup>(2)</sup> as the computational vehicle to test his near wall modeling ideas for convective heat transfer and wall shear stress analysis. He successfully validated the model against a number of turbine stator and rotor experimental databases.

However, the high flow turning in turbine flows leads to significant problems with grid quality in the near wall region. One of the solutions to this problem came from the work of Professor Andrea Arnone, of the University of Florence, and colleagues at NASA LeRC<sup>(3)</sup>. Arnone's approach was to construct a grid that did not require periodicity in the region downstream of the blade trailing edge, thus allowing a "better" grid in the blade passage, near the walls. He called the two versions of this code TRAF2D and TRAF3D. Building on this, Dr. Ali A. Ameri and Arnone<sup>(4,5)</sup> collaborated to adapt Boyle's heat transfer modeling to the TRAF3D code.

In the meantime, Dr. Vijay K. Garg built on the work of Boyle and adapted the Chima code for the analysis of film-cooled turbine blades<sup>(6)</sup>. This was followed by a switch to the TRAF3D code to take advantage of the improved grid near the walls<sup>(7)</sup>. Garg's initial approach for film cooling was to concentrate surface grid points over the exits of the film cooling holes and specify a distribution of in-flow boundary conditions at those points. This worked well if the number of holes was small, but the penalty in additional grid points throughout the computational domain was significant. A new approach to the gridding of complex problems was needed.

This issue was addressed by Dr. Erlendur Steinthorsson and colleagues<sup>(8)</sup>. Steinthorsson revised the TRAF3D flow solver, developing a new code, TRAF3D.MB, that was specially designed to use multiblock grid systems. Under this approach, the computational domain is broken down into a topology of unstructured, connected blocks, and a detailed, structured grid is created within each block. A good description of this new code and a demonstration of its capability to accurately compute heat transfer in a complex flow can be found in the work of Ameri and Steinthorsson<sup>(9)</sup>. In this report they compared their computed heat transfer on the tip and blade surfaces of a rotating turbine blade with experimental data, with very good agreement. Steinthorsson et al<sup>(10)</sup> then extended the multiblock approach to the analysis of internal cooling passage flow, successfully simulating the air flow and convective heat transfer in a complex "branched duct" geometry that included many features typically found in turbine blade cooling passages. At the same time, Rigby et al<sup>(11)</sup> applied the new code to a turbine blade cooling passage model consisting of a 180° turn in a rectangular passage, further demonstrating the advantage of the multiblock grid approach. Rigby et al<sup>(12)</sup> then added the complexity of ribs and bleed holes to the internal passage flow, successfully using the multiblock method to include a detailed grid and calculation in the small holes bleeding coolant flow from the ribbed channel. Garg and Rigby<sup>(13)</sup> further extended the multiblock analysis to show the importance of the within-hole and near-hole physics in relation to heat transfer on a film cooled blade, including in the analysis the coolant plenum, the flow through the film cooling holes, and the external hot gas stream. Heidmann et al<sup>(14)</sup> went another step and considered a blade with two independent coolant plenums and multiple film cooling holes on the leading edge, suction side, and pressure side, including shaped holes. Their predictions of surface heat transfer are being used to guide an experimental effort currently under way at NASA Glenn.

In 1999, the NASA Lewis Research Center name changed to NASA Glenn Research Center, and at that time, the TRAF3D.MB code was renamed the Glenn-HT code. Most recently, a Fortran90/95 version of the Glenn-HT code has been completed, taking advantage of the object-oriented capabilities of Fortran90/95 to produce a code that is more flexible and efficient than the previous version.



In the rest of this report, some code specifics will be presented and examples of the application of the Glenn-HT code to some specific turbine problems will be described, followed by a description of some planned improvements to extend the envelope of applicability for the code.

## RESULTS AND DISCUSSION

### DESCRIPTION OF THE CODE

A description of the code can be found in most of the references already cited. In brief, the Glenn-HT code solves the full compressible Reynolds Averaged Navier-Stokes (RANS) Equations in a rotating coordinate system, using a multi-stage Runge-Kutta scheme to march in pseudo-time, with multi-grid and implicit residual smoothing to accelerate convergence. To handle the complex geometries typically found in turbines, the code uses multi-block grid systems. As currently configured the code uses a  $k-\omega$  turbulence model, and no wall functions. A recent paper by Garg and Ameri<sup>(15)</sup> successfully demonstrated the use of two additional turbulence models in the code. Up to now, the code has been used as a steady-state analysis tool, but the new version has the capability for time-accurate analysis of transient phenomena. One question that needs to be addressed relative to the application to a Turbine Based Combined Cycle (TBCC) Accelerator is whether the analysis of the turbine needs to be done in a transient mode, or is the transient slow enough that the analysis can be done as a series of quasi-steady calculations.

### TURBINE MAINSTREAM & SECONDARY FLOWS

An excellent benchmark data case for heat transfer in a transonic turbine blade cascade has been published by Giel et al<sup>(16,17)</sup>. Figure 2 shows a 3D representation of their measured heat transfer data on the blade and endwall surfaces. Although this was a linear cascade, the combination of thick endwall inlet boundary layers and high blade turning angle produced a highly 3D flow field. Garg and Ameri<sup>(15)</sup> used the Glenn-HT code to compute the same configuration, evaluating three different turbulence models against the data. A sample of the calculations is shown in figure 3. Incidentally, all the experimental aerodynamic and heat transfer data generated for this cascade, and details of the geometry, are available on request on two CDs from the NASA Glenn Research Center.

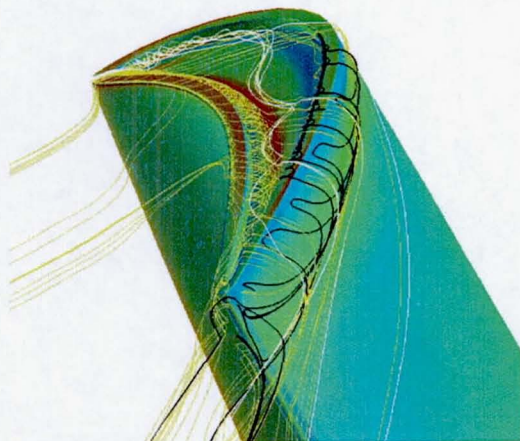


Fig. 4. Glenn-HT Computed Flow Traces and Heat Transfer for a Squealer-Tip Rotor Blade

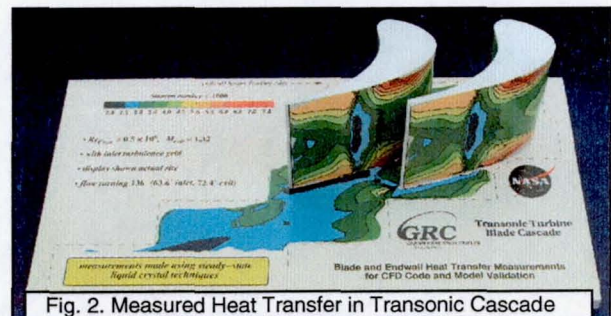


Fig. 2. Measured Heat Transfer in Transonic Cascade

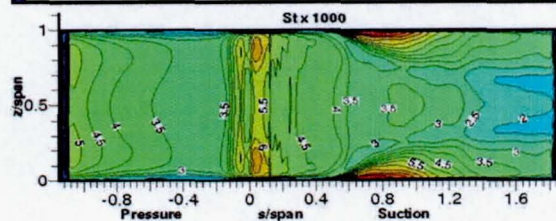


Fig. 3. Glenn-HT Computed Heat Transfer for NASA Glenn Transonic Cascade

The hot gas leakage flow through the gap between the rotor blade tip and the stationary shroud has a very strong influence on losses and durability. Ameri and co-workers have published a number of papers<sup>(9,18-23)</sup> where they compared Glenn-HT calculations of the flow and heat transfer in this region to experimental measurements, with excellent results. Figure 4, from a Glenn-HT calculation, illustrates the complexity of the flow in the tip gap region. This figure uses numerical flow visualization to show the path of the various vortices generated over the blade tip. The surface heat transfer levels are visualized by color, with red being the highest.



Figure 5 shows a comparison of tip heat transfer experimental and computational results. This experiment was run at the General Electric Corporate Research Center, and consisted of a single-blade cascade with contoured side walls and a stagnation line splitter plate. The computational domain had to

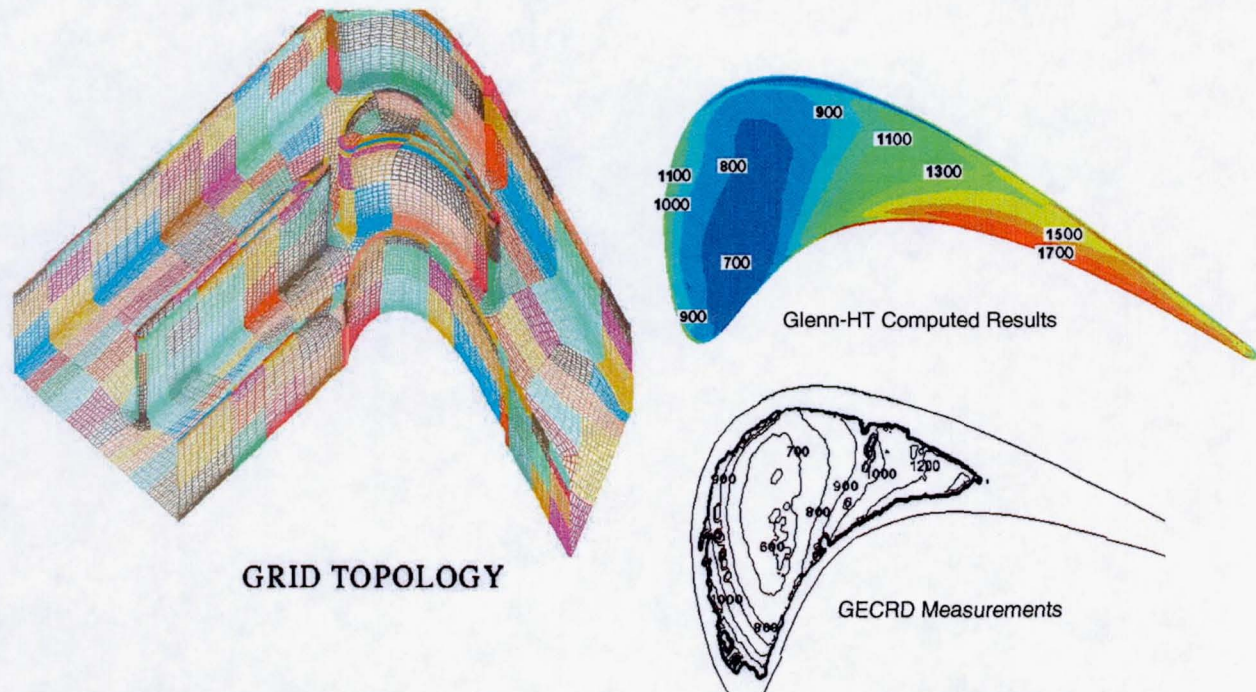


Fig. 5. Computational Grid, and Glenn-HT Computed Blade Tip Heat Transfer Compared to Experimental Data from General Electric Corporate Research Center.

include the entire test section geometry since there were no symmetry planes or periodic boundaries. A comparison of the measured heat transfer on the blade tip in the tip leakage gap and the Glenn-HT calculation showed excellent agreement as seen in Figure 5.

Another tip-clearance flow validation case was conducted by Ameri and Rigby<sup>(22)</sup> where they modeled an actively cooled tip experiment run by Kim and Metzger<sup>(29)</sup>. Figure 6 is a schematic of a typical cooled blade tip, where film-cooling holes on the tip provide a blanket of cooling air to protect the tip region. Kim and Metzger modeled this configuration, isolating the region of the blade pressure side and the adjacent tip region with film cooling holes. The Glenn-HT calculations of Ameri and Rigby showed excellent agreement with the measured film cooling effectiveness.

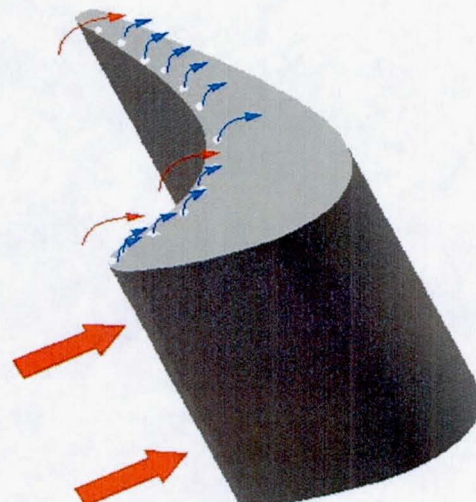


Fig. 6. Typical Rotor Blade with Tip Film-Cooling

#### TURBINE FILM COOLING FLOW

A very effective way to protect turbine blades from the hot gas is to blanket the blade surface with a film of cooling air. However, for analysis purposes, this adds a significant complexity to the problem. A major effort is continuing to validate the Glenn-HT code for the analysis of film-cooled components. Figure 7, from Garg and Rigby<sup>(13)</sup>, compares the experimental results of Camci and Arts<sup>(24)</sup> with Glenn-HT computational results. In this case, periodicity considerations allowed the analysis to be conducted on a representative slice of the tested blade, containing three film cooling holes around the leading edge



stagnation region. Figure 7(a) and (b) show details of the inviscid grid. Note that the coolant plenum and film cooling hole pipes were gridded in detail and included in the calculation. Figure 7(c) shows contours of the computed heat transfer coefficient distribution in the vicinity of the cooling holes, and figure 7(d) compares the computed and measured span averaged heat transfer coefficient distribution, showing excellent agreement.

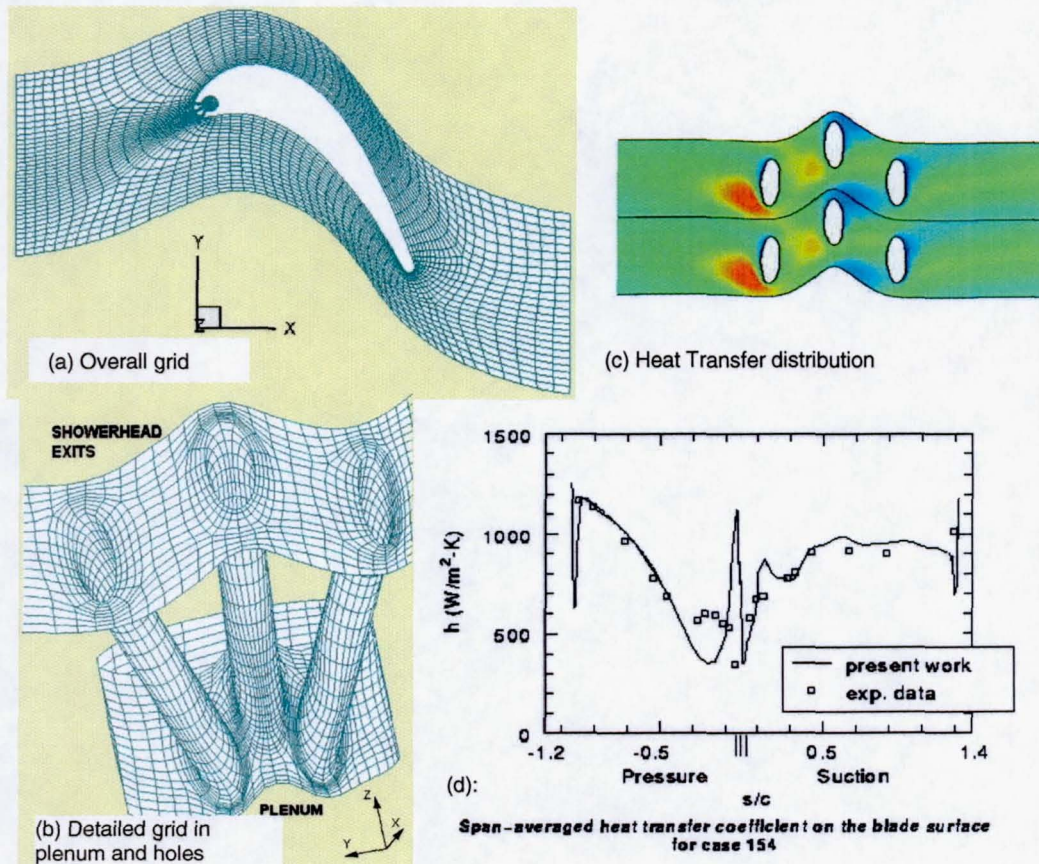


Fig. 7 VKI Blade geometry and heat transfer coefficients

## TURBINE INTERNAL COOLING PASSAGE FLOW

Perhaps the biggest challenge to the Glenn-HT code is to analyze the internal cooling passage flows in a turbine blade or vane. A cartoon cut-away view of a generic cooled blade is shown in figure 8. In general these passages are non-uniform in cross-section, contain turbulators to agitate the flow, contain sharp bends, have local flow injection or ejection, and in the rotor blade, are subject to the body forces generated by rotation. Rigby, Steinthorsson, and co-workers<sup>(11, 25-27)</sup> have concentrated their efforts on these problems, validating the code against a wide range of experimental configurations. Figure 9 shows a comparison of Sherwood Number data from an experiment, Park et al<sup>(28)</sup>, for a square cross section, ribbed, rotating passage with a 180° turn and Glenn-HT calculations by Rigby for the same configuration. This test used the naphthalene sublimation technique to measure mass transfer rather than heat transfer, and the Glenn-HT code was modified to solve the mass transfer

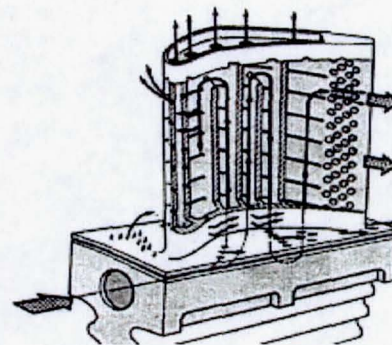


Figure 8, Cut-away view of generic cooled turbine blade.



equations. Note that the color scale is different between the two cases. The computational grid for this problem contained almost 1.5 million cells, distributed in 64 blocks. The agreement is excellent, particularly in the extremely complex flow region in the vicinity of the bend.

Additional validation is underway for some complex real engine cooling configurations under the NASA Ultra Efficient Engine Technology (UEET) Program, but final results are not yet available.

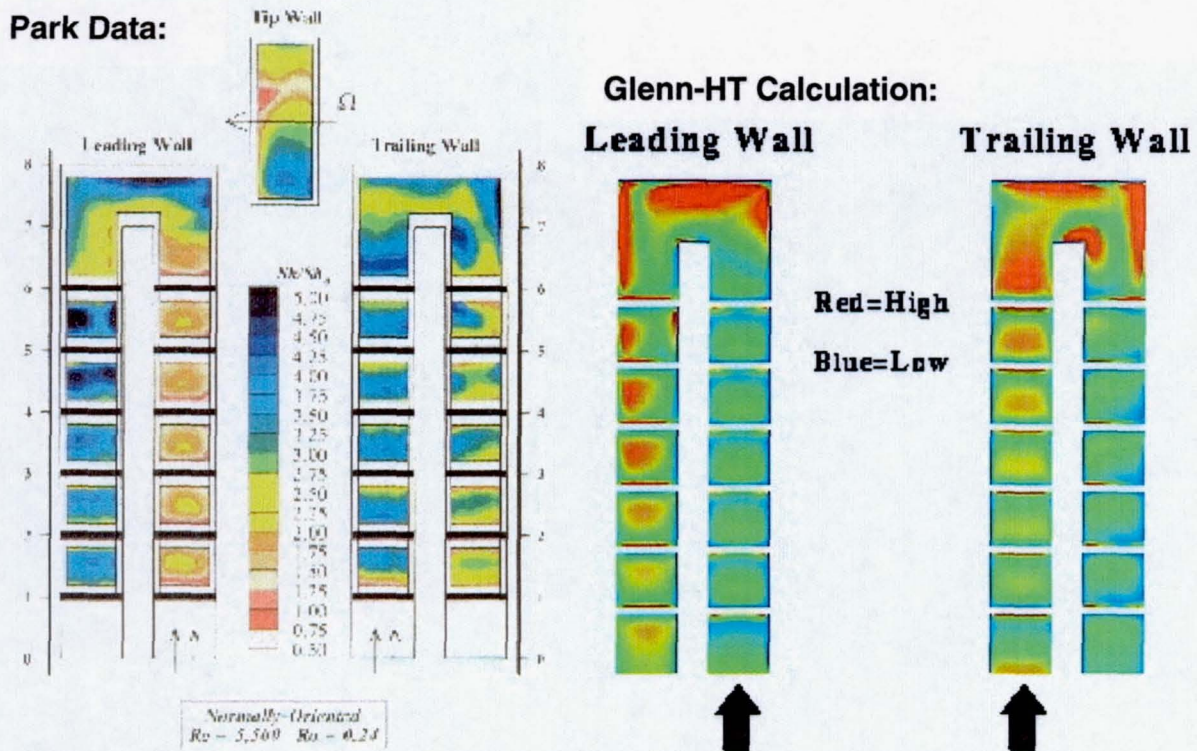


Figure 9, Comparison of Measured and Computed Sherwood Number for a Square, Rotating, Model Cooling Passage with Ribs and a 180° Turn.

### SUMMARY AND CONCLUSIONS

The Glenn-HT code, a three-dimensional (3D), Reynolds Averaged Navier-Stokes (RANS) Computational Fluid Dynamics (CFD) computer code for the analysis of gas turbine flow and heat transfer has been described and examples of its validation and use for complex flow calculations have been presented. The code is unique in the ability to give a highly detailed representation of the flow field very close to solid surfaces in order to get accurate representation of convective heat transfer and viscous shear stresses. An extensive body of literature documenting the code development and validation has been published over the last few years, portions of which were highlighted in this paper. Applications include: a) the main gas path through a turbine, including the fine details of flow and heat transfer in the clearance gap over the blade tip and the complex flow interactions of film cooling flows with the main stream flow; b) the complex flow in internal cooling passages, including turbulators on the walls, injection/ejection of coolant, sharp turns, and rotation; and c) the simultaneous combined interaction of internal and external flows.



In its current form, the Glenn-HT code can be used to analyze, over the broad range of operating conditions, the turbines designed for application to Turbine Based Combined Cycle (TBCC) systems for space launch vehicle propulsion systems. At the same time, code improvements and extensions are being developed to expand the envelope of applicability. The ability to analyze unsteady effects is included in the latest version of the code, but validation efforts are just beginning. There are also efforts underway to adapt the code for conjugate heat transfer calculations, including heat conduction in the solid structures in the analysis. Improvements to the turbulence modeling in the code are being actively pursued, with the ultimate goal to be able to include a Large Eddy Simulation (LES) capability. Efforts are also underway to streamline the grid generation process through automated topology generation.

The current version of the code can be made available for domestic use on request.

## REFERENCES

- (1) Boyle, R. J. "Navier-Stokes Analysis of Turbine Blade Heat Transfer", ASME Paper 90-GT-42, June, 1990. Also published in: ASME Journal of Turbomachinery, July 1991 Vol. 113 pp. 392-403.
- (2) Chima, R. V., "Development of an Explicit Multigrid Algorithm for Quasi-Three-Dimensional Viscous Flows in Turbomachinery", AIAA Paper 86-0032, 1986. Also, NASA TM-87128.
- (3) Arnone, A., Liou, M-S. and Povinelli, L. A., "Navier-Stokes Solution of Transonic Cascade Flows using Non-Periodic C-type Grids", AIAA Journal of Propulsion and Power, v. 8, n. 2, pp. 410-417, March-April, 1992.
- (4) Ameri, A. A. and Arnone, A., "Three Dimensional Navier-Stokes Analysis of Turbine Passage Heat Transfer", AIAA paper 91-2241, June, 1991
- (5) Ameri, A. A. and Arnone, A., "Navier-Stokes Turbine Heat Transfer Predictions Using Two-Equation Turbulence Closures", AIAA paper 92-3067, July, 1992, also NASA TM 105817
- (6) Garg, V. K. and Gaugler, R. E., "Heat Transfer in Film Cooled Turbine Blades," ASME Paper 93-GT-81, May, 1993
- (7) Garg, V. K. and Gaugler, R. E., "Prediction of Film Cooling on Gas Turbine Airfoils," ASME Paper 94-GT-16, June, 1994; also NASA TM-10653
- (8) Steinthorsson, E., Liou, M-S., and Povinelli, L. A., "Development of an Explicit Multiblock/Multigrid Flow Solver for Viscous Flows in Complex Geometries," AIAA-93-2380, 1993. Also, NASA TM 106356.
- (9) Ameri, A. A. and Steinthorsson, E., "Prediction of Unshrouded Rotor Blade Tip Heat Transfer", ASME paper 95-GT-142, June, 1995, also NASA CR 198542
- (10) Steinthorsson, E., Ameri, A. A. and Rigby, David L., "Simulation of Turbine Cooling Flows Using Multiblock-Multigrid Scheme", AIAA Paper 96-0621, January, 1996
- (11) Rigby, David L., Ameri, A. A. and Steinthorsson, E., "Internal Passage Heat Transfer Prediction Using Multiblock Grids and a  $k-\omega$  Turbulence Model", ASME Paper 96-GT-188, June, 1996, also NASA CR-198540
- (12) Rigby, David L., Steinthorsson, E. and Ameri, A. A., "Numerical Prediction of Heat Transfer in a Channel with Ribs and Bleed", ASME Paper 97-GT-431, June, 1997
- (13) Garg, V. K. and Rigby, D. L., "Heat transfer on a Film Cooled Blade - Effect of Hole Physics", ASME Paper 98-GT-404, June, 1998, also NASA/CR-1998-206609. Also published in Int. J. of Heat & Fluid Flow, v. 20 (1999) pp 10-25.



- (14) Heidmann, James D., Rigby, David L. and Ameri, Ali A., "A Three-Dimensional Coupled Internal/External Simulation of a Film-Cooled Turbine vane", ASME Paper 99-GT-186, June, 1999, also NASA/TM-1999-209078, April, 1999, also ASME J. of Turbomachinery, v 122, pp 348-359.
- (15) Garg, Vijay K. and Ameri, Ali A., "Two-Equation Turbulence Models for Prediction of Heat Transfer on a Transonic Turbine Blade", ASME paper 2001-GT-0165, June 2001, NASA/CR—2001-210810
- (16) Giel P. W., Thurman, D. R., VanFossen, G. J., Hippensteele, S. A. and Boyle, R. J., "Endwall Heat Transfer Measurements in a Transonic Turbine Cascade", ASME 96-GT-180, June, 1996, also ASME J. of Turbomachinery, v 120, pp 305-313
- (17) Giel P. W., VanFossen, G. J., Boyle, R. J., Civinskas, K. C. and Thurman, D. R., "Blade Heat Transfer Measurements and Predictions in a Transonic Turbine Cascade", ASME 99-GT-125, June, 1999, also NASA/TM-1999-209296
- (18) Ameri, A. A. and Steinthorsson, E., "Prediction of Unshrouded Rotor Blade Tip Heat Transfer", ASME paper 95-GT-142, June, 1995, also NASA CR 198542
- (19) Ameri, A. A., Steinthorsson, E. and Rigby, David L., "Effect of Squealer Tip on Rotor Heat Transfer and Efficiency", ASME paper 97-GT-128, June, 1997, & ASME Journal of Turbomachinery, October, 1998, vol. 120, no. 4, pp. 753-759.
- (20) Ameri, A. A., Steinthorsson, E. and Rigby, David L., "Effects of Tip Clearance and Casing Recess on Heat Transfer and Stage Efficiency in Axial Turbines", ASME paper 98-GT-369, June, 1998, also NASA/CR-1998-208514, to be published in ASME Journal of Turbomachinery
- (21) Ameri, A. A. and Bunker, R. S., "Heat Transfer and Flow on the First Stage Blade Tip of a Power Generation Gas Turbine, Part 2: Simulation Results", ASME paper 99-GT-283, June, 1999, , also NASA/CR-1999-209151, April, 1999.
- (22) Ameri, A. A. and Rigby, David L., "A Numerical Analysis of Heat Transfer and Effectiveness on Film Cooled Turbine Blade Tip Models", NASA/CR-1999-209165, July, 1999. Presented at 14th ISABE, Florence, Italy, September 5-10, 1999
- (23) Ameri, Ali A., "Heat Transfer and Flow on the Tip of a Gas Turbine Equipped with a Mean-Camberline Strip", ASME paper 2001-GT-0156, June 2001, NASA/CR—2001-210764
- (24) Camci, C. and Arts, T., "Experimental Heat Transfer Investigation Around the Film-Cooled Leading Edge of a High Pressure Gas Turbine Rotor Blade", ASME Journal of Engineering for Gas Turbines and Power, Vol. 107, pp. 1016-1021, 1985.
- (25) Rigby, David L., Steinthorsson, E. and Ameri, A. A., "Numerical Prediction of Heat Transfer in a Channel with Ribs and Bleed", ASME Paper 97-GT-431, June, 1997
- (26) Rigby, David L., "Prediction of Heat and Mass Transfer in a Rotating Ribbed Coolant Passage with a 180 Degree Turn", ASME Paper 98-GT-329, June, 1998, also NASA/TM-1999-208501
- (27) Steinthorsson, E., Ameri, A. A. and Rigby, David L., "Simulations of Heat Transfer in Complex Internal Flows Using a  $k-\omega$  Turbulence Model", AIAA Paper 97-0506, January, 1997
- (28) Park, C. W., Laou, S. C., and Kukreja, R. T., "Heat/Mass Transfer in a Rotating Two-Pass Square Channel with Transverse Ribs", Journal of Thermophysics and Heat Transfer, Vol. 11, 1997.