Three-Dimensional Viscous Flow Analysis for Centrifugal Impellers

C. M. Rhie,* R. A. Delaney,† and T. F. McKain‡
General Motors Corporation, Indianapolis, Indiana

Abstract

A SPACE–MARCHING Navier-Stokes solver is used to simulate three-dimensional viscous flows in centrifugal impellers. This solver employs a finite volume integration procedure for the partially parabolic equations written in arbitrary curvilinear coordinates. The solution is achieved using a multiple-pass forward-marching iteration scheme. The accuracy of the analysis method is verified using impeller intrapassage laser-velocimeter data. The method is demonstrated to predict the jet/wake flows characteristic of centrifugal impellers. The result clearly shows the importance of modeling the blade tip clearance flow accurately.

Contents

Current centrifugal compressor design systems do not account for the complex three-dimensional viscous flow effects occurring in these machines. These effects include rapidly growing endwall and blade surface boundary layers, secondary flows driven by strong cross-stream pressure gradients, tip leakage flows, and a separated flow region near the shroud-suction side corner. These effects limit the attainable impeller pressure rise and stage efficiency. Therefore, it is clear that, in order to achieve substantial gains in component performance, analytical models must be developed to treat these effects accurately. These effects can only be modeled using full three-dimensional viscous flow analyses.

Moore and Moore[1] devised a partially parabolic Navier-Stokes (PPNS) procedure to model realistic impeller geometries and flow conditions; however, they were not able to accurately predict the strong secondary flows and jet and wake structure in the exducer region. Recently, the present authors were successful in obtaining realistic viscous flow predictions for centrifugal impellers using the extended PPNS solver developed by Rhie.[2] This procedure is a generalization of the procedure developed by Pratap and Spalding[3] and solves the PPNS equations in arbitrary curvilinear coordinate systems.

The governing equations were the continuity and time-averaged Navier-Stokes equations written in a rotating reference frame. The constant rothalpy condition is used to calculate the local temperature, and the density is determined using the perfect gas relation. The Reynolds stresses were determined using a two-equation k-ϵ eddy-viscosity turbulence model. The wall function method is used near the wall.

Presented as Paper 84-1296 at the AIAA/SAE/ASME 20th Joint Propulsion Conference, June 11-13, 1984; submitted July 11, 1984; revision submitted Feb. 13, 1985. Full paper available from AIAA Library, 555 W. 57th St., New York, N. Y. 10019. Price: Microfiche, $4.00, hard copy, $9.00. **Remittance must accompany order.**

*Senior Project Engineer, Allison Gas Turbine Division; currently at Pratt & Whitney Aircraft, East Hartford, Connecticut.

†Supervisor, Computational Fluid Mechanics, Allison Gas Turbine Division.

‡Supervisor, Compressor Design, Allison Gas Turbine Division.

To numerically solve the governing differential equations, they were transformed numerically to an arbitrary coordinate system.

$$1/J\left\{\lfloor \rho G_1 \phi \rfloor_\xi + \lfloor \rho G_2 \phi \rfloor_\eta + \lfloor \rho G_3 \phi \rfloor_\zeta\right\}$$
$$= 1/J\left\{\lfloor J\,\Gamma \alpha_1 \phi_\xi \rfloor_\xi + \lfloor J\,\Gamma \alpha_2 \phi_\eta \rfloor_\eta\right.$$
$$\left. + \left\{\lfloor J\,\Gamma \alpha_3 \phi_\zeta \rfloor \zeta + S\right\}\right\} + S^\phi$$

The equations were solved on the blade-conforming coordinate system. Invoking the partially parabolic flow assumption, the diffusion term in the primary flow direction, $\lfloor J\,\Gamma\alpha_3\phi_\zeta \rfloor_\zeta$, is negligible. This assumption is not necessary in the present case since the dependent flow variables are stored into three-dimensional arrays. However, this term is neglected to save computational effort due to its negligibility in all impeller passage flows.

The solution is marched in the primary flow direction on cross-stream planes beginning from the prescribed condition on an initial data plane. The scheme combines the finite volume integration technique for the momentum and turbulence scalar transport equations and a two-step pressure correction procedure to enforce global and local mass conservation. The momentum and turbulence scalar transport equations are solved using an alternating direction implicit scheme. Second-order-accurate central differences are used in the cross-stream plane and first-order-accurate backward differences are used in the primary flow direction.

Computations were made for the radial discharge impeller recently tested by Eckardt.[4] The impeller was run at the design operating condition defined by the following parameters: rotational speed = 14,000 rpm, flow rate = 5.31 kg/s, stage total pressure ratio = 2.08, and stage aerodynamic efficiency = 88.8%.

The computations were carried out on a sheared-type grid. Constant area annular sections were added upstream and downstream for the impeller so that circumferentially uniform boundary conditions could be applied at the inlet and exit. A coordinate stretching procedure was used to cluster points in the high-gradient near-wall and blade tip regions. The grid was comprised of 15 points in the circumferential direction, 19 points in the spanwise direction, and 50 points in the primary flow direction. Four of the 19 points in the spanwise direction were packed between the blade tip and shroud surface.

Care was taken to model the actual impeller boundary conditions as closely as possible. In the absence of information on the upstream velocity profiles, the conditions on the boundary were established by calculating the developing flow in a straight annular duct upstream of the boundary. The length of the duct, determined through numerical experimentation, was equal to ten times the blade span at the leading-edge span at the leading edge. The conditions along the downstream boundary were determined by extrapolating the solution from

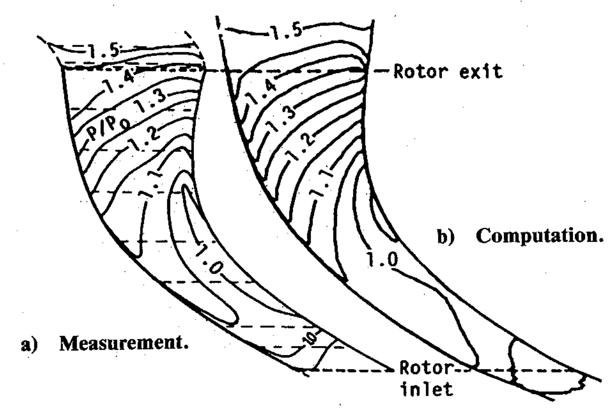
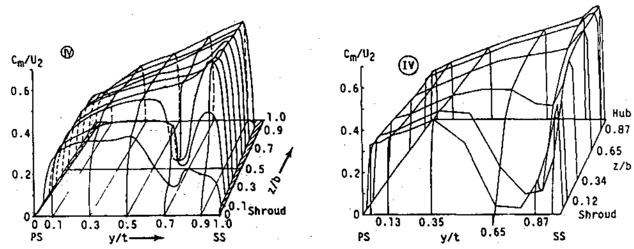


Fig. 1 Static pressure distribution at the shroud.



a) Measurement. b) Computation.

Fig. 2 Flow condition within the impeller at plane IV.

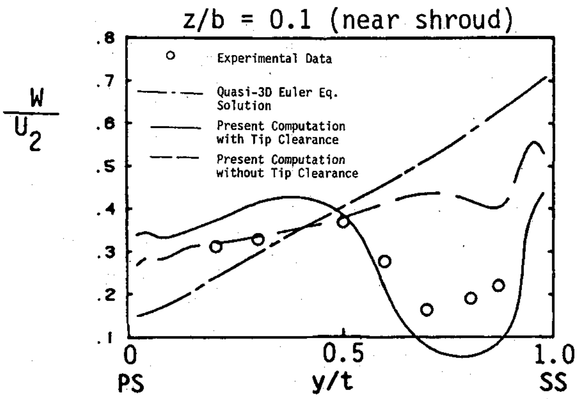
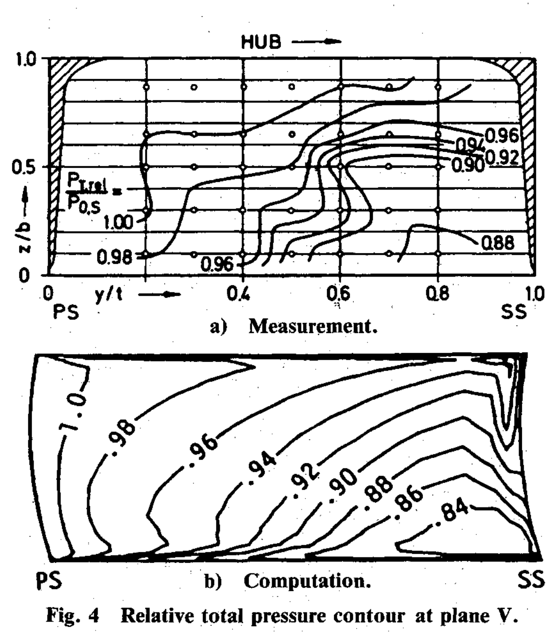


Fig. 3 Relative velocity profiles at plane IV.



a) Measurement.

PS b) Computation. SS

Fig. 4 Relative total pressure contour at plane V.

neighboring upstream points. Periodic conditions were enforced along lateral boundaries upstream and downstream of the impeller and in the tip clearance region. Consistent with information reported by Eckardt, the blade tip clearance was specified as 0.7% of the blade height at plane I, 1.5% at plane III, and 1.9% at plane V. Plane I was at 7.5%, plane II at 57.5%, and plane III at 100% meridional locations measured along the shroud.

The solutions were obtained using 250 global iterations of the space-marching procedure and took approximately 1 h of CPU time on an IBM 3033 computer.

Contour plots of the calculated and measured shroud surface static pressure distributions are compared in Fig. 1. The plots show excellent correlation over the entire surface. In the exducer region, the contours indicate strong radial and blade-to-blade gradients characteristic of centrifugal impeller flows.

Comparison of the predicted and experimental data at measurement plane IV (87.5% meridional location) is shown in Fig. 2. Of particular note is that the analysis simulated the characteristic jet and wake structure. The discrepancies between the measured and predicted profiles are attributed to the use of a relatively coarse computational mesh.

Predictions of the relative velocity distributions made with and without tip clearance are presented in Fig. 3 at tip location ($z/b = 0.1$) at this measurement plane to demonstrate the importance of modeling the tip clearance flow. The predictions with tip clearance flow show better agreement with the experimental data. It is clear from this comparison that the tip clearance flow triggers earlier wake development and increases the size of the wake region. Also shown in the figure is the result from a quasi-three-dimensional inviscid Euler equation solution obtained by Schuster and Schmidt-Eisenlohy.[5] Poor agreement is shown. This is expected because the quasi-three-dimensional inviscid solver cannot predict the wake and jet structure which is strongly influenced by the viscous secondary flow.

Figure 4 shows the predicted and experimental relative total pressure (normalized by the isentropic total pressure) contours at measurement plane V. In view of the reported ±2% uncertainty on the measured values, these contours show remarkably good agreement. Both plots clearly show the high-loss wake region in the corner between the blade suction and shroud surfaces.

In the present work, all of the characteristic jet/wake structures in a centrifugal impeller were predicted using a multiple-pass space-marching procedure to solve the partially parabolic Navier-Stokes equations. It was quantified that the tip clearance flow has a dominant effect on the wake formation process. The present solution approach is efficient for subsonic impeller flows as long as the leading-edge elliptic effect remains relatively small.

References

[1]Moore, J. and Moore, J. G., "Three-Dimensional Viscous Flow Calculation for Assessing the Thermodynamic Performance of Centrifugal Compressors, Study of the Eckardt Compressor," *Proceedings of the AGARD Meeting on Centrifugal Compressors, Flow Phenomena and Performance*, Brussels, Belgium, May 1980.

[2]Rhie, C. M., "Basic Calibration of a Partially-Parabolic Procedure Aimed at Centrifugal Impeller Analysis," AIAA Paper 83-0260, Jan. 1983.

[3]Pratap, V. S. and Spalding, D. B., "Fluid Flow and Heat Transfer in Three-Dimensional Duct Flows," *International Journal of Heat and Mass Transfer*, Vol. 19, Sept. 1976, p. 1183.

[4]Eckardt, D., "Detailed Flow Investigations within a High Speed Centrifugal Compressor Impeller," *Journal of Fluids Engineering, Transactions of ASME*, Ser. F, Vol. 98, Sept. 1976, 390-402.

[5]Schuster, P. and Schmidt-Eisenlohy, U., "Flow Field Analysis of Radial and Backswept Centrifugal Compressor Impellers, Part 2: Comparison of Potential Flow Calculations and Measurments," ASME Publication on Performance Prediction of Centrifugal Pumps and Compressors, 1979, pp. 87-95.