JOURNAL OF PROPULSION AND POWER
Vol. 21, No. 5, September–October 2005

Semiviscous Method for Compressor Performance Prediction

Magdy S. Attia*
Embry-Riddle Aeronautical University, Daytona Beach, Florida 32114

The performance prediction of the modern axial compressor is a vital step in the design and analysis process. Not only are accurate flow and efficiency predictions necessary, but also reliable radial and axial work distributions predictions are needed to verify the design, and more importantly, uncover problems. Furthermore, an accurate and consistent predictive system can minimize the need for testing, resulting in substantial savings. Although traditional S1S2 type systems are fast, they are limited in the transonic and subsonic regimes and are not very reliable often failing at or near the end walls. Multirow computational fluid dynamics (CFD) has come a long way, but also has its share of issues such as meshing, turbulence models, and mixing plane formulations. Furthermore, setup and turnaround time for a multistage compressor make multirow CFD less than efficient and often requiring "expert" users. This paper presents a simple, physically sound, reliable, and efficient method for predicting axial compressor performance using a combination of single-row CFD and streamline curvature (Throughflow) codes. The method has been developed and tested on several industrial multistage compressors of varying sizes and loadings. In each case, the prediction was well within the measurement accuracy.

Nomenclature

P = pressure
V = flow velocity in a nonrotating frame of reference
W = flow velocity in a rotating frame of reference
η = adiabatic efficiency
ξ = loss coefficient
π = total to total pressure ratio

I. Introduction

PREDICTING compressor performance is probably the most important step in the design process of a gas turbine engine. Traditionally, companies are reluctant to proceed with a truly clean sheet design. Extensive computations are conducted and compared to older, successful designs. Deviation from previous experience and previously successful designs is scrutinized, and only tweaks are allowed. Expensive tests are then conducted to verify and validate the new design. It is fair to say that manufacturing and mechanical robustness considerations have played a role in slowing down the evolution of the compressor. However, the aerodynamic uncertainty is still the driver behind the overall design philosophy.

It can be argued that the more devastating aerodynamic design mistakes would result in an axial mismatch. A poor estimate of the flow function entering a stage would, at the very least, lead to poor performance, and will, almost always, necessitate a redesign. The true consequences of such a redesign far exceed the high cost or the missed deadline. They extend to the psyche of the organization as a whole and often contribute to an overall shying away from most innovations as they are quickly deemed "risky".

Cumpsty[1] presents a comprehensive overview of the evolution of performance analysis. It is generally agreed upon that the Throughflow code is used and referred to by many as the main tool for compressor performance prediction. Denton,[2] Hearsey,[3] Hale et al.,[4] and others have presented slightly differing approaches for dealing with streamline curvature equations. However, all approaches need losses and deviation, in one form or another, as an input to accurately compute the aerodynamics and thermodynamics of the stream section. Cumpsty[1] offers a detailed description, in the form of a flowchart, for analysis using streamline curvature (Throughflow) codes. Step 4 of the outlined process[1] recommends the use of blade-to-blade analysis or correlation to determine adequate values for losses and deviation.

Work in the field of determining adequate values for losses and deviation for use in a streamline curvature code has been extensive and often quite clever yielding remarkably realistic results. Adkins and Smith,[5] Gallimore and Cumpsty,[6] and Gallimore,[7] tackled the issue of spanwise mixing in an effort to improve the general predictive capability of the typical Throughflow code, with good results. Boyer and O'Brien[8] proceeded further, and in a more specialized manner, to better model shock losses in the typical Throughflow code. Inevitably, and without exception, intensive massaging is needed before implementation into the streamline curvature code. Generalizations, assumptions, and simplifications are employed to marry the new principles to the streamline equation. As a result, blockage factors are still extensively utilized to deal with end-wall effects, and formulations are categorized as either for purely transonic or purely subsonic airfoils, with no formulation covering both regimes. Often, research has either focused on losses or deviation, not both. The identity and character of the individual airfoil is what is important here, and that has not yet been successfully modeled. The use of internal stations [calculation stations between the leading edge (LE) and trailing edge (TE) of the airfoil] can be quite complimentary, but is of secondary importance.

Focus has then shifted to computational methods. Fully viscous, three-dimensional, Reynolds-averaged Navier–Stokes approximations indeed offer the greatest fidelity in modeling[9] and have since become invaluable. Multistage computational fluid dynamics (CFD) has also gained in popularity, particularly after the introduction of Adamczyk's average passage scheme,[10] facilitating the modeling of various, previously ignored, phenomena such as hub leakage effects.[11] However, these methods are not without limitations. Mesh size, turbulence models, and mixing plane formulations are a few of the issues facing CFD users. Consequently, it has almost become a requirement that expert status be achieved before using multirow CFD effectively.

The preceding research motivated the work described in this paper, which effectively marries the two approaches, three-dimensional CFD and streamline curvature, capturing the best of both worlds: the fidelity of the CFD and the robustness and quickness of the Throughflow. At the same time all correction factors such as blockage and loss redistribution are eliminated.

Presented as Paper 2004-3412 at the AIAA/ASME/SAE/ASEE 40th Joint Propulsion Conference and Exhibit, Ft. Lauderdale, FL, 11–15 July 2004; received 20 October 2004; revision received 17 January 2005; accepted for publication 31 January 2005.

*Assistant Professor, Department of Aerospace Engineering, 600 South Clyde Morris Boulevard. Member AIAA.

II. Methodology

The methodology described here deals with the modeling of viscosity, or the effects of viscosity, in a historically inviscid code, namely, a streamline curvature code or Throughflow. A Throughflow code is inherently an excellent bookkeeper of axial and radial work distributions. However, it relies heavily on input values of losses and air angles to calculate the aerodynamics and thermodynamics of each stream section. The objective of the semiviscous method is to mine such values from single blade row (single blade passage), steady, three-dimensional CFD calculations and impose them on the streamline curvature code. The procedure is as follows.

Blockage is completely eliminated from the Throughflow solution, and so is any remixing. The initial solution of the Throughflow is usually an estimate, or the design intent if available. It is highly recommended that the initial solution contain little or no end-wall effects to avoid biasing the process. The mass flow rate is allowed to vary, and convergence is achieved when the pressure ratio imposed on the compressor at the operating point under consideration is met. Care is taken in the setup of the Throughflow input as to ambient conditions, inlet guide vane (IGV) position (if any), humidity, and bleed mass flow rates. Additionally, a sensitivity study was conducted to assess the benefit of including internal stations and yielded no noticeable difference. However, as some Throughflow codes are equipped to calculate aerodynamic forces for structural reasons, it was noted that internal stations were beneficial for that purpose, especially for transonic airfoils. In general, Throughflow codes have limits on the total number of computational stations, and liberal use of internal stations, particularly in multistage compressors, will quickly exhaust this limit.

CFD jobs for all airfoils are initialized and launched with preliminary boundary conditions imposed by the initial Throughflow converged solution just alluded to. The CFD mesh chosen here was structured, H-grid, single block, and measuring approximately 106,000 nodes. The chosen size was a result of various sensitivity studies conducted to determine an adequate size for the problem, such that good accuracy is achieved with a single size mesh for all airfoils within a reasonable turnaround time. The mesh extends from the suction surface of one airfoil to the pressure surface of the adjacent airfoil. For unshrouded rotors, three radial computational planes were placed in the tip clearance. Because of the desire to use a single grid mesh, blade tips were modeled as cusps with the radial mesh planes at and near the tips in close proximity to better simulate flat tips. The CFD employed circumferentially uniform boundary conditions, and the Baldwin–Lomax turbulence model was chosen after several studies. Results of the CFD runs, namely, losses and air angles, are implemented as input to the Throughflow code. The first convergence of the Throughflow solution under these conditions yields an updated set of boundary conditions (BCs). The new BCs are then employed in the next round of three-dimensional CFD for all airfoils, for a much more realistic solution. The new losses and air angles are again imposed on the Throughflow as input, and a second calculation is converged. The resulting streamline solution is then used to evaluate the performance of the compressor.

A. Loss Modeling

Loss could be modeled by various means. The recommended scheme is to use the loss coefficient as defined by

$$\xi = \Delta P_0 \Big/ \tfrac{1}{2}\rho V^2$$

where P_0 and V are the total pressure and flow velocity relative to the row, respectively. Specifying the total pressure or temperature values instead of losses and air angles (as some Throughflow codes will allow) tends to have a restrictive effect in that it hinders the code's ability to redistribute work axially and, hence, is not recommended. As shown in Fig. 1, the loss is modeled as accurately as possible. Although the typical Throughflow mesh will employ a small number of streamlines to model the flow (in this case 21 equally spaced streamlines), the typical three-dimensional mesh will employ a larger number of radial gridlines and thus produce a level of detail that cannot be mimicked by the Throughflow.

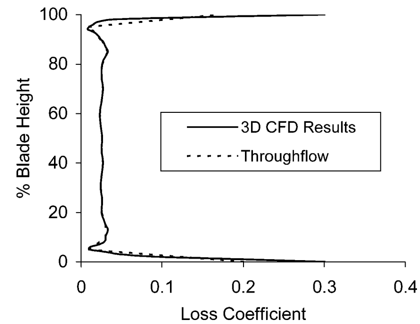

Fig. 1 Loss modeling scheme employed by the semiviscous method.

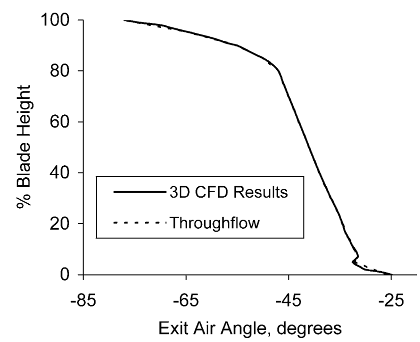

Fig. 2 Exit air angle modeling scheme employed by the semiviscous method (for a typical rotating blade).

Engineering judgment is called upon to produce the best possible representation. The question of the extent of the "mimicking" will be addressed in the section titled Multistage Considerations.

B. Air Angles Modeling

In a similar fashion as the loss-modeling scheme, exit air angles are modeled. An iterative procedure can be employed here to guarantee matching at the exit of the three-dimensional CFD computation domain. Whereas air angles are historically input at the TE of the airfoil in a Throughflow code, comparison (to guarantee matching) is performed at the exit of the CFD computation domain, which is customarily the LE of the downstream airfoil. Allowance for the "straightening" of the flow must be made. An example of air angle modeling is shown in Fig. 2.

C. Multistage Considerations

It is evident from careful examination of the figures that the typical Throughflow code can lead to difficulty with convergence issues should the user continuously employ such high values for loss coefficient and air angle at the end walls. The cumulative effect of high losses and turning angles causes a rapid increase in temperature coupled with a proportional drop in total pressure across the stages. This will result in a dangerous drop in axial velocity and a breakdown of the velocity triangle at the end wall, or rather, the ability of the Throughflow code to resolve the triangles at the end walls.

Initially, it was suggested that resolution of the convergence issue might be achieved via the use of truncated models, such as modeling a 15-stage compressor in three five-stage segments, for example. Although this approach was successful, it only offered a marginal improvement. Furthermore, the truncated-model approach rendered the automation of the method significantly more complicated, particularly for compressors with a large number of stages. It was decided to only use this approach for special cases, for example, where the accurate prediction of radial distribution of thermal properties was needed for comparison with measurement data.

The chosen approach was to manually redistribute the larger values of losses and air angles starting at the rear of the compressor and moving upstream. Care should be taken in order not to compromise the shape, or trend, of the original (three-dimensional CFD) loss or exit air angle radial distribution. The process aims to impose the least amount of redistribution necessary to converge the Throughflow. Figure 3 shows an example of exit air angle redistribution at the last rotor of a multistage compressor without significantly compromising the shape or trend of the CFD. In this case, the air angle closest to the end wall is highest and continues to decrease smoothly as we move toward the midspan, following the same trend as the CFD. The redistribution is halted at 70% of the blade height. Essentially, the user here is asked to perform a "manual" mixing operation to aid in the convergence of the Throughflow. Loss redistribution follows a similar scheme.

In summary, the objective of the method is to capture air angles and losses from the CFD and depict them in the Throughflow as accurately as possible. If, after using all available means (such as varying the relaxation factors), convergence of the Throughflow remains unachievable, redistribution is employed as a last resort. Several sensitivity studies were conducted to ascertain the impact of such redistribution on the overall prediction. First, it was found that such redistributions were not at all necessary for compressors with moderate number of stages (10 or less), that is, it was possible to converge the Throughflow solution using very large exit air angles and losses. Second, radial redistribution had minimal to no impact on overall values such as flow rate and adiabatic efficiency, if they followed the preceding example. Additionally, it had no impact on the axial distribution of work. Total pressure distribution at the compressor exit was only slightly impacted, and total temperature distribution was only moderately impacted. As a general rule, when necessary, the redistributions were deemed acceptable.

D. Aerodynamic Instability Considerations

Perhaps the most useful feature of this method is its ability to capture aerodynamic instabilities such as corner stalls, shock-induced separations, exceptionally large tip vortices, and successfully model these disturbances in the Throughflow code, and propagate their impact. It is in this regard that the semiviscous method far exceeds other more conventional methods in modeling fidelity.

Figure 4 shows the scheme by which a large hub corner stall (separation) detected by the three-dimensional CFD is captured and modeled by the Throughflow. Upon convergence of the solution, the effect of the separation is propagated across the compressor, and its impact is manifest in the form of decreased stage and overall efficiency in a proportional manner and altered axial work distribution. three-dimensional CFD can then be executed to further quantify the impact on adjacent rows, for example. Capturing and modeling such an event would not be possible using the more conventional methods. It can be argued that the individuality of the airfoil is largely ignored by the more conventional methods, and thus they are more susceptible to overlooking significant aerodynamic issues. Furthermore, highly specialized geometric features such as bowed stators or end bends, which cannot be modeled by the typical Throughflow code, are automatically accounted for by the semiviscous method.

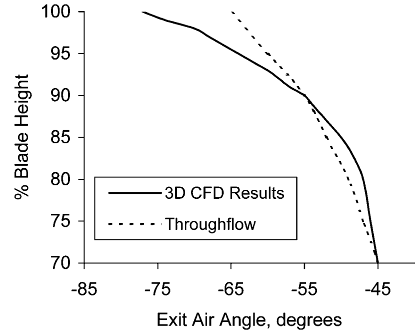

Fig. 3 Exit air angle redistribution at the rear of the compressor for management of convergence issues.

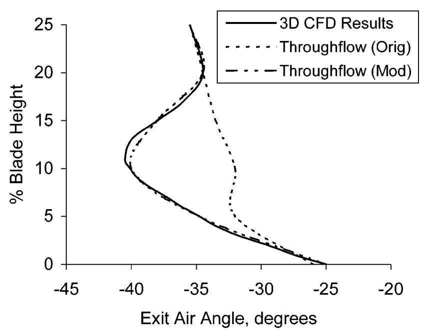

Fig. 4 Exit air angle modification capturing a large corner stall (for a rotating airfoil).

III. Validation

To validate the method, analysis was conducted for several different compressors employing different airfoil shapes ranging from NACA-65 to double circular arc (DCA) to highly specialized custom diffusion airfoils for transonic applications. Aspect ratios varied from 1.4 to 3.25, airfoil span varied from 0.6 to 0.05 m, and stage total-to-total pressure ratios varied from 1.7 to 1.05. Operating conditions also varied, from design point to fully open IGV; an 89% referred speed case was also analyzed. At the extreme, a far off-design case was studied; a depressed inlet pressure condition led to an effective Reynolds number that was 1/15th of the design point Reynolds number. Tip gaps varied from 0.3% of span to 2.9% span in the rear of the compressor. In many instances, three-dimensional CFD indicated that some of the airfoils, particularly on the older frames, were operating with a small corner stall. For the depressed inlet pressure case, a few airfoils suffered from substantial separation near the hub, which could not be resolved by multirow CFD, but the single row CFD employed here was robust enough to converge. This was modeled in the Throughflow, as already outlined. In one instance, shocks with high inlet Mach number formed on the suction surface of a transonic rotor. The CFD result was recalculated using a matrix of varying mesh sizes, for further refinement, vs various turbulence models. While the fine mesh refined the shock with clearer resolution, the overall loss and deviation changed little from the initially chosen setup. Two of the cases are presented next.

A. Case 1: 16-Stage Compressor

This 16-stage compressor consisted of DCA airfoils for the front stages followed by modified NACA-65 airfoils. Reliable shop test data existed at an IGV position of +4 deg (closed); overall total-to-total pressure ratio for this operating point was approximately 16:1.

The initial effort to analyze this compressor was started by using the traditional S1S2, where S2 is the streamline curvature (Throughflow) code and S1 refers to two-dimensional blade-to-blade solution on the stream surface. This two-dimensional solution is inviscid with superposition of a simple boundary-layer calculation. The S1S2 approach has an advantage over empirical methods and methods that employ correlation, because the geometry of the airfoil is considered and accounted for. However, the two-dimensional nature of the S1 was a contributor to the poor and inaccurate modeling of end-wall flows. Additionally, because of convergence issues, the hub and tip streamlines were not modeled, but rather the next streamlines were calculated, and the radial distribution is "extrapolated" without the benefit of three-dimensional effects. Not all of the streamlines were analyzed by the S1, but only five of the 21 for each blade row at 5, 25, 50, 75, and 95% blade height. As a consequence of all of the preceding, the S1S2 analysis produced a virtually flat total pressure distribution at the exit, grossly overestimated the adiabatic efficiency, and underestimated the flow. Attempts to reallocate blockage, or force more loss to the end wall-regions, were abandoned, as they were neither repeatable, nor successful. Rather, a method that incorporates end-wall effects in a physics-based and repeatable manner, which also eliminates the need for all correction factors such as blockage, as well as eliminating the need for radial mixing, was called for.

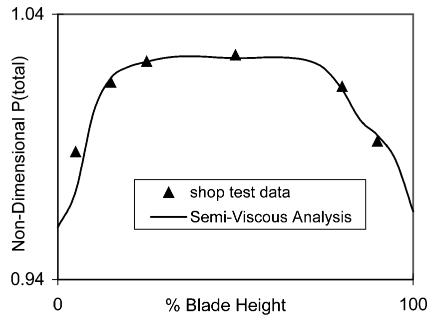

Fig. 5 Comparison of semiviscous prediction vs total pressure measurements at the inlet to the last stator of a 16-stage compressor.

Initially, the Throughflow code was converged using the design intent radial distribution of thermodynamic and aerodynamic values. The first run of CFD calculations converged rather easily using the setup just outlined. The results showed, as expected, that several stators at the rear of the compressor were operating with small regions of corner stall (rearward of 90% axial chord and up to 10% span). End-wall phenomena such as high losses and deviation were captured and implemented in the Throughflow. The Throughflow code was converged, and the mass flow rate was allowed to float. Convergence was not obtained easily, and some redistribution was necessary, as already outlined. However, the redistribution was only needed rearward of stage 14, and the trend was maintained.

The reconverged Throughflow results, after one iteration, were remarkably different. End-wall effects were clearly visible, as would be expected. The boundary conditions, to be given to the next round of CFD, had been altered to reflect the reality of the situation. The next round of CFD calculations was conducted using identical mesh and problem setup. As anticipated, incidence had increased at the hubs, and with it losses, and the magnitude of the separations. The results of the CFD calculations were then imported into the Throughflow for a second iteration.

The result of the Throughflow solution after only two iterations was remarkable. Mass flow rate and overall total-to-total adiabatic efficiency agreed to within 0.3% of the measurement. Figure 5 shows a comparison between the calculated and measured total pressure at the inlet of the last (16th) stator. Excellent agreement is evident to well within the measurement uncertainty band.

B. Case 2: 19-Stage Compressor at Extreme Off-Design ($Re = 1/15 Re_{\mathrm{ref}}$)

Building on the results of case 1, several more compressors were analyzed using the semiviscous method. Operating conditions varied widely, as well as geometry and loading levels. A particularly challenging situation was presented after a test was conducted for a 19-stage compressor, which employed highly specialized custom diffusion airfoils, had a highly transonic front stage, and had substantial tip gaps in the rear.

The size of this compressor made it impossible to find a motor that could supply enough power to operate it at its design point. The test was devised such that only a fraction of the power would be needed. The approach adopted to accomplish this was to depress the inlet total pressure from 14.7 to 1 psia. The test was successfully conducted, and the compressor did reach its design pressure ratio of 31:1. However, because of the obvious disparity between the test Reynolds number and the design intent Reynolds number, losses were excessive. Hence, the efficiency measured was well below the design point efficiency. The semiviscous method was used to calculate the performance of the compressor at test conditions and later recompute the would-be performance at normal operating conditions.

The process was as outlined for case 1. However, in this case, with the mass flow rate being 1/15th of the design intent, separations and corner stalls were persistent and often quite large. A problem was quickly identified near the front of the compressor. To try and model the problem with more fidelity, multirow CFD (three-dimensional steady with multiple frames of reference) was

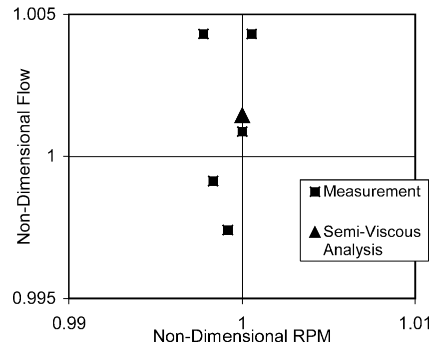

Fig. 6 Comparison of semiviscous prediction vs flow measurements for a 19-stage compressor at extreme off-design conditions.

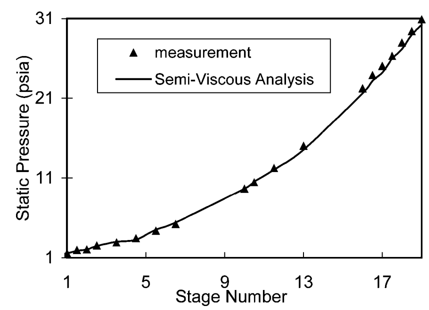

Fig. 7 Comparisons of semiviscous prediction vs tip static-pressure measurements for a 19-stage compressor at extreme off-design conditions.

employed first for a five-row domain ($2\frac{1}{2}$ stages) and later for a three-row domain ($1\frac{1}{2}$ stages). In both instances, significant effort was expanded to try and converge the multirow solution with no success. Multirow CFD was then abandoned. The semiviscous method was able to easily converge a single-row CFD run for this airfoil using the Baldwin–Lomax turbulence model. Separation extended forward to 50% chord at the hub and diminished at 30% span. Losses and deviation were quite extensive. Applying these values to the Throughflow required loss and deviation redistribution at the rear, as already mentioned, to achieve convergence. Two iterations were conducted, as for case 1 before, and the mass flow rate predicted matched that measured (Fig. 6).

Figure 7 shows a comparison between predicted and measured tip static pressures for the entire length of the compressor. As can be clearly seen, the aerodynamic instabilities near the front of the compressor are well defined and accurately depicted by the semiviscous analysis. A subsequent analysis of the same compressor at design-point operation uncovered a problem at the same location. The airfoils in question were redesigned resulting in substantial savings and rendering an otherwise-useless test quite effective.

IV. Conclusions

A method for analyzing and predicting the performance of compressors, at design or off-design operating conditions, has been presented. The method couples the fidelity and accuracy of single-row CFD with the robust, easy-to-use Throughflow code via unambiguous means. It employs no correction factors; it is based on sound physics and is proven to be repeatable. Simple CFD setup, along with single size mesh, was used for all airfoils studied, resulting in an easily automated process. Several sensitivity studies were conducted during the development phase to justify the choices made. Validation against measurements has been demonstrated for two entirely different cases. In each case the prediction was well within the measurement uncertainty.

References

[1]Cumpsty, N. F., *Compressor Aerodynamics*, Longman Scientific and Technical, Longman Group, Essex, England, UK; also Wiley, New York, 1989, pp. 98–110 and 129–131.

[2]Denton, J. D., "Throughflow Calculations for Transonic Axial Flow Turbines," *Journal of Engineering for Power*, Vol. 100, April 1978, p. 212.

[3]Hearsey, R. M., "Program HT0300 NASA 1994 Version," The Boeing Co., Doc. No. D6-81569TN, St. Louis, Vols. 1 and 2, Feb. 1994.

[4]Hale, A. A., Davis, M. W., and Kneile, K. R., "Turbine Engine Analysis Compressor Code: TEACC—Part I: Technical Approach and Steady Results," AIAA Paper 94-0148, Jan. 1994.

[5]Adkins, G. G., Jr., and Smith, L. H., Jr., "Spanwise Mixing in Axial-Flow Turbomachines," *Journal of Engineering for Power*, Vol. 104, Jan. 1982, p. 97.

[6]Gallimore, S. J., and Cumpsty, N. A., "Spanwise Mixing in Multistage Axial Flow Compressors: Part I—Experimental Investigation," *Journal of Turbomachinery*, Vol. 108, July 1986, p. 3.

[7]Gallimore, S. J., "Spanwise Mixing in Multistage Axial Flow Compressors: Part II—Throughflow Calculations Including Mixing," *Journal of Turbomachinery*, Vol. 108, July 1986, p. 10.

[8]Boyer, K. M., and O'Brien, W. F., "An Improved Streamline Curvature Approach for Off-Design Analysis of Transonic Compression Systems," ASME Paper GT-2002-30444, June 2002.

[9]Denton, J. D., and Dawes, W. N., "Computational Fluid Dynamics for Turbomachinery Design," *Proceedings of the Institution of Mechanical Engineers, Journal of Mechanical Engineering Science, Part C*, Vol. 213, No. C2, 1999, pp. 107–124.

[10]Adamczyk, J. J., Mulac, R. A., and Celestina, M. L., "A Model for Closing the Inviscid Form of the Average Passage Equation System," *Journal of Turbomachinery*, Vol. 108, Oct. 1986, p. 187.

[11]Shabbir, A., Adamczyk, J. J., Strazisar, A. J., and Celestina, M. L., "The Effect of Hub Leakage Flow on Two High Speed Axial Flow Compressor Rotors," American Society of Mechanical Engineers, Paper 97-GT-346, June 1997.