45th AIAA Aerospace Sciences Meeting and Exhibit
8 - 11 January 2007, Reno, Nevada

AIAA 2007-519

Interaction of Turbomachinery Components in Large-scale Unsteady Computations of Jet Engines

Georgi Kalitzin [*], Gorazd Medic [†], Edwin van der Weide [‡], and Juan J. Alonso [§]
Stanford University, Stanford, CA 94305-4035

I. Introduction

The objective of the Stanford ASC project[2] is to develop a framework able to perform multi-disciplinary, integrated simulations on massively parallel platforms.[3,4] This paper focuses on the turbomachinery computation and, in particular, on the physics of interaction of different turbomachinery components in the engine. Typical flow features such as tip and horse-shoe vortices as well as blade wakes will be discussed for these multi-component turbomachinery simulations.

The compressor and turbine of a modern turbofan engine, Figure 1, typically have two counter-rotating concentric shafts to allow for different rotational speeds of their components as well as a reduction of net torque. The low-pressure parts rotate at a lower rate than the high-pressure components. Typical rotation rates are 5,000 to 7,000 RPM for the former and 15,000 to 20,000 RPM for the latter. The compressor and turbine themselves consist of a series of rotors and stators for which the blade counts are normally chosen such that no sector periodicity occurs. Combined with the inherently unsteady nature of turbomachinery flows due to the motion of the rotors, the full wheel geometry needs to be considered in a time accurate numerical simulation of the flow.

The computational requirements for such a simulation are severe. The high-pressure compressor (HPC) alone consists of 5 stages (rotor/stator combinations) and 50 to 200 blade passages per stage. Since approximately a million nodes are required per blade passage to obtain a grid-converged Reynolds-Averaged Navier-Stokes (RANS) solution, the computational mesh for a full wheel HPC simulation contains 500 million to 1 billion nodes. The turbine consists of less stages due to the the favorable pressure gradient. However, a full wheel simulation still requires 150 million to 300 million nodes. The spatial mesh is to be integrated in time for 2,000 to 10,000 time steps, based on the estimate that 50 to 100 time steps are needed to resolve a blade passing, to remove the transient effects. Alone the full wheel unsteady HPC simulation will require 20 to 40 million CPU hours on today's fastest computers. Adding the low-pressure compressor (LPC), fan as well as high- and low-pressure turbine (HPT and LPT, respectively), the computational requirements are far beyond what is currently affordable for practical applications and therefore approximations are used to reduce the computational costs.

The most widely-used industrial practice for solving turbomachinery problems is the mixing plane assumption.[1] A circumferential averaging of the flow variables is applied at the interface between rotor and stator. These average quantities are then imposed as upstream and downstream values for the following and preceding blade rows respectively and a steady-state computation can be performed for both the rotor and the stator. Due to this averaging and the periodicity assumption only one blade passage needs to be simulated per blade row, independently of the blade counts. Although this assumption models the mean effect of the rotor/stator interaction, all the unsteady information is lost due to the averaging.

An alternative approach used to perform an unsteady simulation is to chose a periodic sector of, for example 20^o, where the blade counts are changed such that the full wheel can be split into 18 sections and periodicity conditions can be used. The pitch and chord of the blades then need to be adjusted to preserve the flow blockage. Because of these changes it is clear that only approximate information can be obtained

---

[*] Research Associate, Center for Integrated Turbulence Simulations, AIAA Member
[†] Research Associate, Center for Integrated Turbulence Simulations, AIAA Member
[‡] Research Associate, Department of Aeronautics and Astronautics, AIAA Member
[§] Associate Professor, Department of Aeronautics and Astronautics, AIAA Member

Copyright © 2007 by the American Institute of Aeronautics and Astronautics, Inc. All rights reserved.

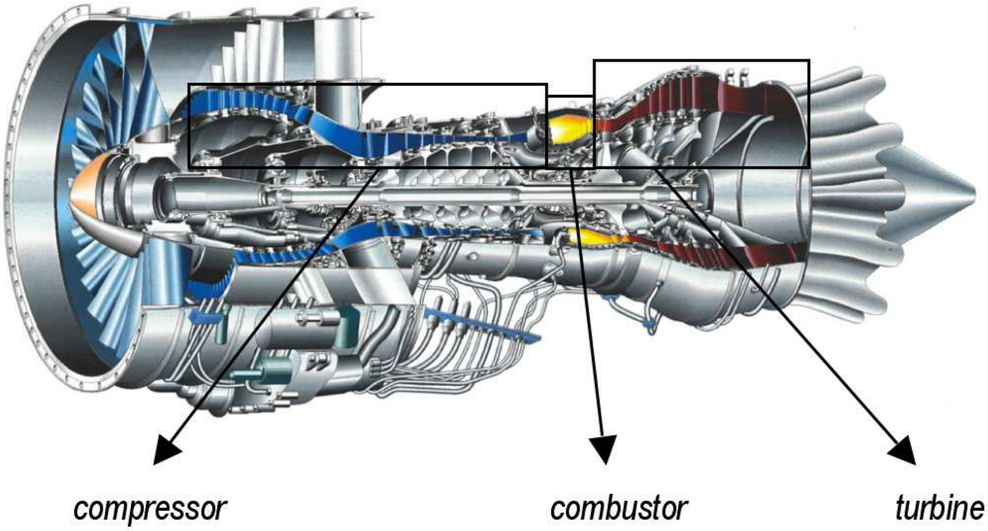

Figure 1. Schematics of an aircraft jet engine.

from an unsteady sector simulations. Nevertheless, as we intend to show, interesting results about the interactions between different turbomachinery components can still be obtained.

II. Flow solver: SUmb

Despite the progress made in unstructured grid technology during the last 10 years, the quality of solutions obtained on structured grids is still superior compared to their unstructured counterparts. This is especially true for high Reynolds number RANS simulations where high aspect ratio cells must be used to capture the anisotropic flow phenomena. In combination with the fact that it is relatively straightforward to create multi-block structured grids for the geometries used in the turbomachinery components of a jet engine, all the compressor and turbine simulations carried out within the Stanford ASC program use multi-block structured grids.

The flow solver performing these simulations is SUmb, which has been developed under the sponsorship of the Department of Energy Advanced Strategic Computing (ASC) Initiative. SUmb solves the compressible Euler, laminar Navier-Stokes and RANS equations on multi-block structured meshes; SUmb is a parallel code, suited for running on massively parallel platforms.[5]

SUmb can be used to solve steady-state problems, unsteady problems (with moving geometries) and time periodic problems, for which a Fourier representation is used for the time derivative leading to a coupled space-time problem.[6,7] A number of turbulence models is available, including k-ω,[8] Spalart-Allmaras[9] and v^2-f.[10] To reduce the grid resolution requirements in the near wall region adaptive wall functions are used.[11,12]

For the discretization of the inviscid fluxes either a central difference scheme augmented with artificial dissipation[13] or an upwind scheme in combination with Roe's approximate Riemann solver[14] is used. The viscous fluxes are computed using a central discretization. All the results presented in this paper are obtained with a second order cell-centered discretization. The second order implicit time integration scheme is used for all unsteady computations. The resulting nonlinear system is solved using the dual time-stepping approach.[15] The convergence is accelerated via a standard geometrical multi-grid algorithm in combination with an explicit multi-stage Runge-Kutta scheme.

III. Results

This section presents some preliminary results for unsteady flow computations of both the fan/compressor and the turbine components of a typical aircraft jet-engine. Both the compressor and the turbine grids have the same topology consisting of an O-grid around the blade and an H-type grid in the passage. The tip gap regions of the rotors have also been resolved. The initial spacing for both the turbine and the fan/compressor grids corresponds to an average y^+ value of 60. The total number of cells in a passage is approximately 350,000 for the turbine. For the compressor grid the number of cells per passage differs per blade row, but on average approximately 500,000 cells per passage are used. The subsonic inflow and outflow boundary conditions correspond to the regular cruise condition of the engine. For all results shown the turbulence is modeled using the k-ω model using wall functions.

A. Fan / compressor: 20^o sector computations

For the 20^o sector simulations the blade counts are changed so that the full wheel can be split into 18 sections and periodicity conditions can be used. The pitch and chord of the blades are adjusted to preserve the flow blockage. The steady solutions obtained with the mixing plane assumption are used as initial conditions for the unsteady computations. The computational grid consists of 204 blocks and 57 million cells (the second level grid has about 7 million cells); 1,200 processors were used on the LLNL ALC machine for the fine grid computations and 400 processors were used for the second level grid. The physical time step is chosen so that a blade passage of the blade row with the highest blade count is resolved with 50 time steps. For the low-pressure components (fan, LPC and LPT) this corresponds to 2,700 time steps per revolution, while for the high-pressure components (HPC and HPT), which rotate at a higher speed, 6,300 time steps per revolution must be taken.

The computational geometry is shown in Figure 2. The airflow through the fan is split into a part that goes through the LPC and a part that goes through the bypass. The latter is not included in these computations; an exit bypass pressure boundary condition is specified.

The fan / compressor simulation is far more challenging to compute than the flow through the turbine due to the large adverse pressure gradient. The pressure at the high-pressure compressor exit is about 30 times larger the pressure at the fan inlet, see Figure 3 (note that a log scale is used for the pressure plotted here). The flow is pushed through the compressor by the work transmitted from the rotating blades to the fluid and an accurate modeling of the boundary layers is crucial to achieve the designed pressure ratio. In the computations, the flow tends to reverse its direction as soon as large pockets of separation occur in the passage.

As a consequence, a special algorithm had to be developed to create an initial flow field. For this, the flow is first computed on a very coarse grid - here we use the 3rd multigrid grid, i.e. a grid that is 64 times coarser than the finest grid. The computation is carried out by first slowly raising the rotational speed of the wheels while keeping the pressure low at the exit, and then by increasing the pressure once the full rotational speed was achieved. Although the 3rd multigrid level does not resolve the boundary layers at all, it is quite remarkable that even on this grid a pressure increase of about 65% of the design pressure ratio could be achieved. The solution is then interpolated on the next finer grid and the procedure of slowly raising the back pressure is continued. Performing computations on successively refined grids also gives insight into the grid convergence and the order of accuracy of the approach.

Results after 10000 time steps computed on the second level grid are presented in Figures 4 and 5; the turbulent kinetic energy is plotted in an axial cross-section just downstream of the fan and on a surface at a certain radial distance from the hub (corresponding to mid-span in the LPC), respectively. Clearly visible are the wakes of the blades in the fan/LPC, as well as the tip vortex from the fan blades. The large wakes from the fan blades are preserved over many stages of the low pressure compressor. These wakes may amplify turbulence in the LPC creating acoustic noise. The interaction of these wakes with the downstream stators and rotors and its effects on the efficiency and flow capacity in the low-pressure compressor are currently investigated in detail. Specifically, the comparisons are being made to the current industry practices to compute the flow through the low pressure compressor, which usually don't model these unsteady wakes originating from the fan blades.

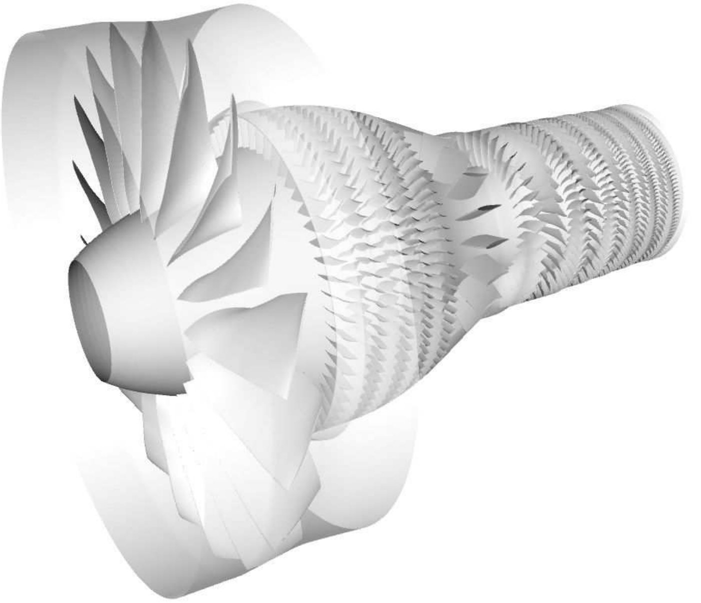

Figure 2. PW 6000 fan/compressor with casing.

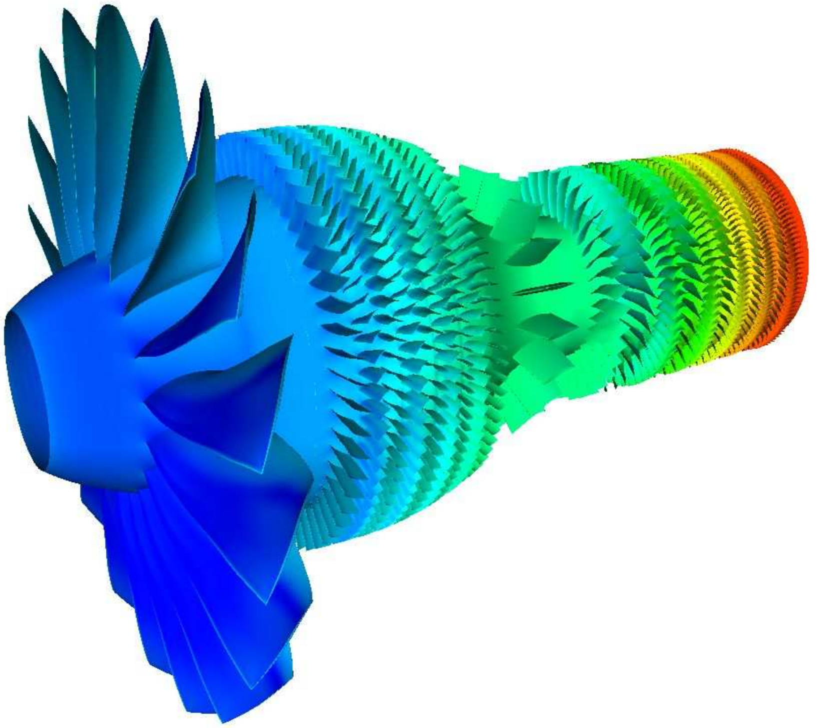

Figure 3. Pressure distribution plotted in log scale.

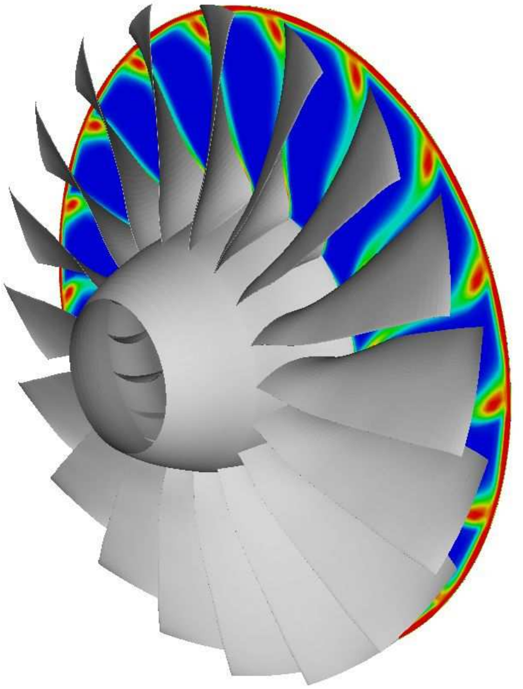

Figure 4. Instantaneous turbulent kinetic energy distribution in an axial plane showing wakes and the tip vortex.

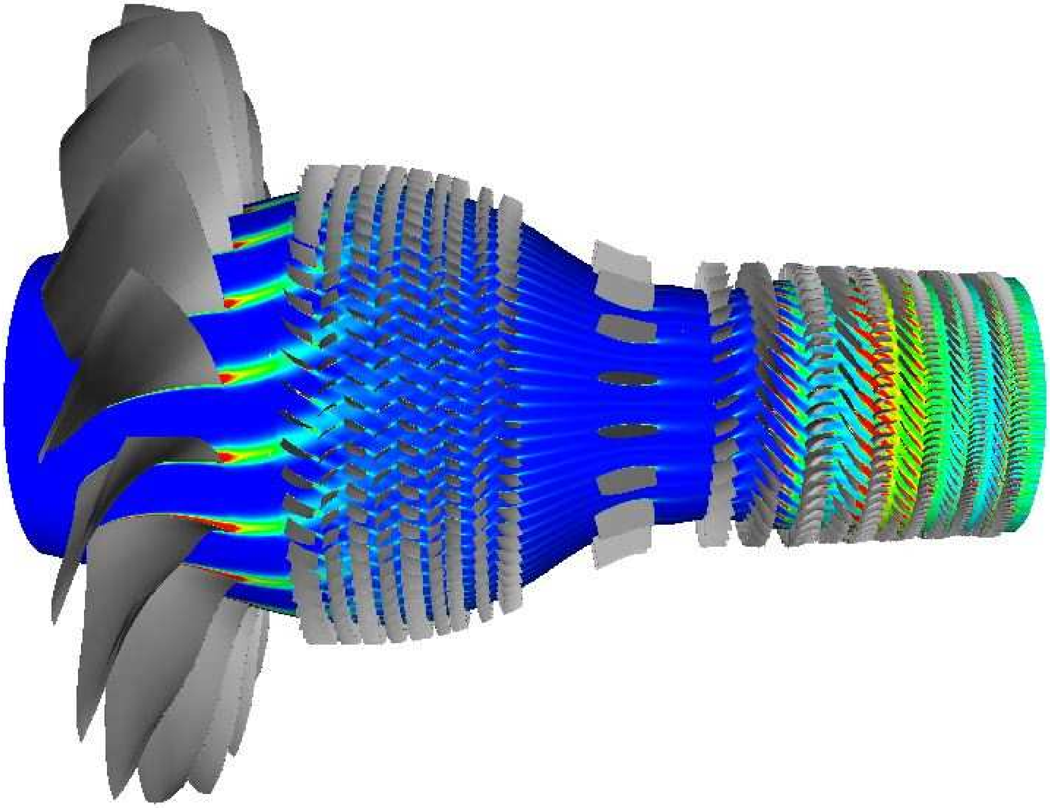

Figure 5. Instantaneous turbulent kinetic energy distribution in a radial plane showing wakes.

B. Turbine: full-wheel computations

Due to the smaller number of blade rows in the turbine, the computational cost for the unsteady simulation of the full wheel turbine are smaller than for the compressor. Consequently most of the attention for the full wheel simulations has been paid to the high-pressure turbine augmented with one stage of the low pressure part. The computational grid consists of 496 blocks and 88 million cells. The disk space needed to store this grid in double precision is 2.1 GBytes. In addition, at least 6 Gbytes of disk space is needed to store the set of independent variables in the solution file. However, if more information is stored in the solution file this number increases. As for the 20^o sector simulation, one revolution of the HPT is resolved with 2,700 time steps.

This case has been run on the LLNL ALC machine on either 300, 600, 1,200 or 1,800 processors, depending on the available resources; SUmb writes the solution in a single file and a restart can be made on a different number of processors due to the fully integrated parallel preprocessor. The solution has been advanced 600 time steps starting from the mixing plane solution. A comparison with the mixing plane and unsteady 20^o sector simulation is shown in Figure 6. The quantity displayed is entropy on a surface located half-way between the hub and the casing. For the unsteady simulations an instantaneous distribution is shown. The difference between the steady computation using the mixing plane assumption and the unsteady simulations is evident. The unsteady nature of the flowfield combined with the different blade counts leads to a strong interaction (both downstream and upstream) between the rotor and stator. This information is lost due to the circumferential averaging and results in a completely different picture of the flow field. The difference between both unsteady simulations is not as clear, but subtle differences can be distinguished. Due to the rescaling of the blade counts in the 20^o sector simulation the wake patterns of the preceding blade rows differ from the true geometry and hence the interaction between the blade rows shows different frequencies.

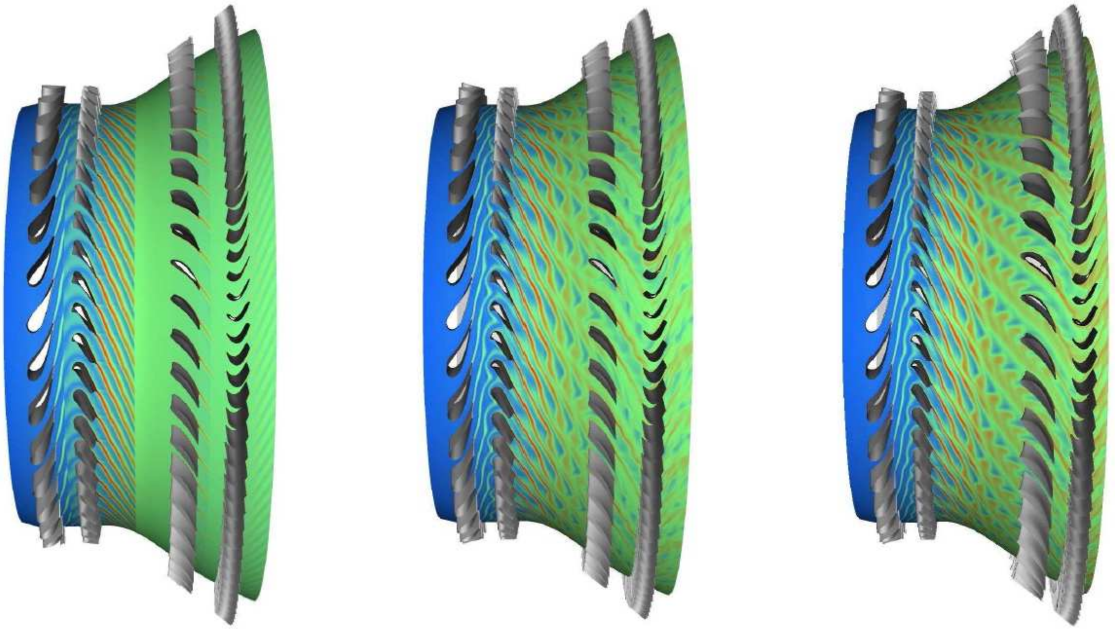

Figure 6. Entropy distributions for simulations of the first two stages of a modern turbine. Steady solution with the mixing plane approximation (left), unsteady solution for a 20^o sector with scaled geometry (middle) and unsteady solution for the full wheel (right).

IV. Conclusions

An unsteady simulation of the flow in the turbomachinery provides a significant improvement compared with the widely-used industrial practice of steady flow computation using the mixing plane assumption. The unsteady nature of the flowfield leads to a strong interaction between the rotor and stator wakes, and on a larger scale, it leads to a strong interaction between the components of the turbomachinery. In the mixing plane assumption, this information is lost due to the circumferential averaging resulting in a completely different picture of the flow field.

The unsteady flow simulation carried out for a 20^o sector of a scaled fan/low-pressure compressor/high-pressure compressor ensemble has shown that the wakes of the fan blades are preserved over many stages of the low pressure compressor. The interaction of these wakes with the downstream turbomachinery stages effects the performance of the low-pressure compressor and may well contribute to noise generation. Although, as shown for the turbine, the unsteady flow results for a full wheel and a 20^o sector simulation differ significant less when compared to the steady flow computation using the mixing plane assumption, the effect of rescaling of the blade counts remains to be investigated. The different blade counts effects the wake pattern and interaction frequency. Nevertheless, unsteady computations of a scaled sector provide a relatively cheap way towards understanding the physics of unsteady turbomachinery flows.

V. Acknowledgments

We thank the U.S. Department of Energy for the support under the Advanced Simulation and Computing Program. We also thank Pratt & Whitney for providing the engine geometry, computational meshes, helpful comments and discussions.

References

[1] Denton, J. and Singh, U., "Time Marching Methods for Turbomachinery Flows," *VKI LS 1979-07*, 1979.

[2] Center for Integrated Turbulence Simulations, "Annual ASC Report," http://cits.stanford.edu, 2006.

[3] Schlüter, J., Wu, X., van der Weide, E., Hahn, S., and Alonso, J., "Integrated LES- RANS of an Entire High- Spool of a Gas Turbine," *AIAA paper 06-0897*, 44th AIAA Aerospace Sciences Meetings and Exhibit, Reno, NV, January 2006.

[4] Medic, G., You, D., Kalitzin, G., van der Weide, E., Alonso, J., and Pitsch, H., "Integrated RANS-LES Computations of an Entire Gas Turbine Jet Engine," *AIAA*, 45th AIAA Fluid Dynamics Conference and Exhibit, Reno, NV, January 2007.

[5] E. van der Weide, G. Kalitzin, J. S. and Alonso, J., "Unsteady Turbomachinery Computations Using Massively Parallel Platforms," *AIAA paper 06-0421*, 44th AIAA Aerospace Sciences Meetings and Exhibit, Reno, NV, January 2006.

[6] Gopinath, A. and Jameson, A., "Time Spectral Method for Periodic Unsteady Computations over Two- and Three-Dimensional Bodies," *AIAA paper 05-1220*, AIAA 43rd Aerospace Sciences Meeting and Exhibit, Reno, NV, January 2005.

[7] van der Weide, E., Gopinath, A., and Jameson, A., "Turbomachinery Applications with the Time Spectral Method," *AIAA paper 05-4905*, 35th AIAA Fluid Dynamics Conference and Exhibit, Toronto, Ontario, June 2005.

[8] Wilcox, D., "Reassesment of the scale-determining equation for advanced turbulence models." *AIAA Journal*, Vol. 26, 1988, pp. 1299–1310.

[9] Spalart, P. and Allmaras, S., "A one-equation turbulence model for aerodynamic flows." *La Recherche Aerospatiale*, Vol. 1, pp. 1–23.

[10] Durbin, P., "Separated flow computations with the k-ε-$\overline{v^2}$ model." *AIAA Journal*, Vol. 33, 1995, pp. 659–664.

[11] Kalitzin, G., Medic, G., Iaccarino, G., and Durbin, P., "Near-wall behavior of RANS turbulence models and implications for wall function," *Journal of Computational Physics*, Vol. 204, 2005, pp. 265–291.

[12] Medic, G., Kalitzin, G., and van der Weide, E., "Adaptive wall functions with applications." *AIAA paper 06-3744*, 36th AIAA Fluid Dynamics Conference and Exhibit, 2006.

[13] Jameson, A., Schmidt, W., and Turkel, E., "Numerical solution of the Euler equations by finite volume methods using Runge Kutta time stepping schemes," *AIAA paper 81-1259*, 1981.

[14] Roe, P., "Fluctuations and Signals - A Framework for Numerical Evolution Problems," *Numerical Methods for Fluid Dynamics*, Academic Press, 1982.

[15] Jameson, A., "Time Dependent Calculations Using Multigrid, with Applications to Unsteady Flows Past Airfoils and Wings," *AIAA paper* 91-1596, June 1998.