AIAA 2003–0000

Integration of RANS and LES Flow Solvers for Simultaneous Flow Computations

J. U. Schlüter, H. Pitsch, S. Shankaran, J. J. Alonso and P. Moin

Center for Turbulence Research (CTR) & Aerospace Computing Lab (ACL)

Stanford University, Stanford, CA 94305-3030

Abstract for the 41st AIAA Aerospace Science Meeting

January 6–9, 2003/Reno, NV

For permission to copy or republish, contact the American Institute of Aeronautics and Astronautics 1801 Alexander Bell Drive, Suite 500, Reston, VA 20191–4344

INTEGRATION OF RANS AND LES FLOW SOLVERS FOR SIMULTANEOUS FLOW COMPUTATIONS

J. U. Schlüter[*], H. Pitsch[†], S. Shankaran[‡], J. J. Alonso[§] and P. Moin[¶]
Center for Turbulence Research (CTR) & Aerospace Computing Lab (ACL)
Stanford University, Stanford, CA 94305-3030

INTRODUCTION

NUMERICAL flow predictions have reached a level, where they provide valuable information for the design of single components of complex flow systems. The computation of a whole flow system, e.g. a complete aero-engine gas turbine, is out of reach of today's flow solvers due to the optimization of flow solvers for certain flow problems. While most aspects of design problems can be addressed by separate component computations, only the simultaneous computation of all parts can guarantee the proper prediction of multi-component phenomena, such as – in the example of a gas turbine engine – compressor/combustor instability and combustor/turbine hot-streak migration.

In the compressor and the turbine sections of a gas turbine, the flow solver has to be able to handle the moving blades, model the wall turbulence, and predict the pressure and density distributions properly. This can be done accurately by a flow solver based on the Reynolds-Averaged Navier-Stokes (RANS) approach. Several flow solvers optimized for turbo-machinery have been developed and have been in use in research and industry for years.

On the other hand, the flow in the combustion chamber is governed by large scale turbulence, chemical reactions, and the presence of fuel spray. Experience shows that these phenomena require an unsteady approach[8]. Hence, the use of a Large Eddy Simulation (LES) flow solver is desirable. LES flow solvers have been developed mainly in universities over the recent years and are currently finding their way into industrial applications.

Therefore, a promising strategy to perform full aero-thermal simulations of gas-turbine engines is the use of a RANS flow solver for the compressor section, a LES flow solver for the combustor, and again a RANS flow solver for the turbine section (Fig. 1). In most cases, RANS and LES flow solvers have been developed and optimized separately over several years. An attempt to merge these flow solvers into a single code is tedious at best and error-prone due to the different approaches. Instead, a more promising approach is to build on the experience of already existing flow solvers and allow them to run simultaneously. The flow solvers exchange information at overlapping interfaces in order to capture the interactions of different components.

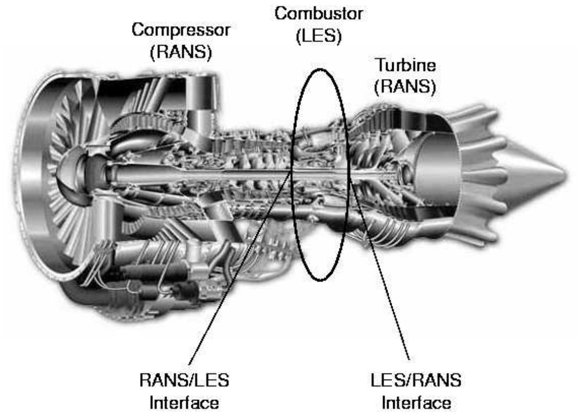

Fig. 1 Gas turbine engine

However, the choice to use multiple flow solvers introduces problems in the level of flow physics – especially the formulation of the boundary conditions – and in efficient program implementation. To explore these challenges the current work reports the coupling of a RANS and a LES flow solver. In order to assess the accuracy and the applicability of this concept to complex flows a swirl flow is computed with a combination of LES and RANS. A swirl flow has been chosen, since this kind of flow can be found in a typical gas turbine combustion chamber and, due to its sensitivity to boundary conditions, is an excellent test-case to proof the importance of integrated flow simulations.

FLOW SOLVERS

The RANS Flow Solver

The unsteady Reynolds Averaged Navier-Stokes equations are solved using a cell-centered discretization on arbitrary multiblock meshes. The solver is parallelized using domain decomposition, an SPMD (Single Program Multiple Data) strategy, and the Message Passing Interface (MPI) Standard.[9]

---

[*] Research Associate, CTR
[†] Research Associate, CTR
[‡] ..., ACL
[§] Assistant Professor, ACL
[¶] Professor, CTR

Copyright © 2002 by the American Institute of Aeronautics and Astronautics, Inc. No copyright is asserted in the United States under Title 17, U.S. Code. The U.S. Government has a royalty-free license to exercise all rights under the copyright claimed herein for Governmental Purposes. All other rights are reserved by the copyright owner.

The solution procedure is based on efficient explicit modified Runge-Kutta methods with several convergence acceleration techniques such as multigrid, residual averaging, and local time-stepping. These techniques, multigrid in particular, provide excellent numerical convergence and fast solution turnaround. Turbulent viscosity is computed from a $k-\omega$ two-equation turbulence model. The dual-time stepping technique[10–12] is used for time-accurate simulations that account for the relative motion of rotors and stators as well as other sources of flow unsteadiness.

The multiblock strategy facilitates the treatment of arbitrarily complex geometries using a series of structured blocks with point-to-point matching at their interfaces. This point-to-point matching ensures global conservation of the flow variables. The structure of the mesh is specified via a connectivity file which allows for arbitrary orientations of the blocks. Two layers of halo cells are used for inter-block information transfer and an efficient communication scheme is implemented for the halo cell data structures. The load of each processor is balanced on the basis of a combination of the amount of computation and communication that each processor performs. Communication of halo cell values is conducted at every stage of the Runge-Kutta integration and in every level of the multigrid cycle in order to guarantee fast convergence rates. A general and parallel efficient procedure has been developed to handle the inter-blade row interface models, for multi-stage turbomachinery simulations.

The LES Flow Solver

The LES flow solver chosen for this work, is an code developed at the Center for Turbulence Research by Pierce and Moin[5]. The flow solver solves the filtered momentum equations with a low-Mach number assumption on an axi-symmetric structured mesh. A second-order finite-volume scheme on a staggered grid is used.[1]

The subgrid stresses are approximated with an eddy-viscosity approach. The eddy viscosity is determined by a dynamic procedure[24]. The low computational costs associated with a structured LES flow solver allows to explore the novelty of LES boundary conditions from RANS computations in depth.

LES BOUNDARY CONDITIONS

One of the biggest challenges in the coupling of LES and RANS flow solvers is the formulation of LES boundary conditions. The different mathematical formulations of both approaches lead to the following problem: unsteady LES boundary conditions have to be generated which fulfill the statistical properties of the time-averaged solution delivered by the RANS flow solver. Even if an unsteady RANS computation is assumed, the time-step of the unsteady RANS computation can be larger than the LES time-step by several orders of magnitude. The LES boundary condition then has to correspond to the ensemble-average delivered by the RANS computation.

The problem of specifying inflow conditions for LES from upstream RANS data is similar to specifying LES inflow conditions from experimental data, which is usually given in time-averaged form, and has therefore been investigated in some detail. A method that has been successfully applied in the past, is, for instance, to generate a time-dependent database for inflow velocity profiles by performing a separate LES simulation in which virtual body forces are applied to achieve the required time-averaged solution[6]. However, since the RANS flow solver delivers unsteady ensemble-averaged velocity profiles, a generation of such a data-base is impossible. Instead, the statistical properties of the data-base have to be altered, so that they match the properties of the RANS solution in the following way:

$$u_{i,\mathrm{LES}}(t) = \underbrace{\bar{u}_{i,\mathrm{RANS}}}_{I} + \underbrace{(u_{i,\mathrm{DB}}(t) - \bar{u}_{i,\mathrm{DB}})}_{II} \cdot \underbrace{\frac{\sqrt{\overline{u'^2_{(i)}}}_{\mathrm{RANS}}}{\sqrt{\overline{u'^2_{(i)}}}_{\mathrm{DB}}}}_{III} \tag{1}$$

with RANS denoting the solution delivered by the RANS computation and DB properties delivered by the database. Term II computes the velocity fluctuation of the database, while term III scales the fluctuation to the actual value needed. When added to term I a meaningful unsteady inlet condition is recovered. In order to keep corrections small, the generated inflow data-base should have statistical properties close to the actual prediction by the RANS flow solver.

The problem of imposing RANS velocity profiles at the outlet of LES computations can be dealt with using virtual body forces.[7] In a specified volume near the outlet of the LES domain a virtual body force is employed:

$$F_i(\boldsymbol{x}) = \frac{1}{\tau_{\mathrm{F}}}(\bar{u}_{i,\mathrm{RANS}}(\boldsymbol{x}) - \bar{u}_{i,\mathrm{LES}}(\boldsymbol{x}))\,, \tag{2}$$

with $\bar{u}_{i,\mathrm{RANS}}$ the solution of the RANS flow solver computed in an overlap region between LES and RANS domain, and $\bar{u}_{i,\mathrm{LES}}$ a time-average of the LES solution over a trailing time-window. This body force ensures, that the velocity profiles at the outlet of the LES domain fulfill the same statistical properties as the velocity profiles in an overlap region computed by a RANS simulation downstream. This makes it possible to take upstream effects of downstream flow alterations into account.

RANS BOUNDARY CONDITIONS

RESULTS

In order to proof the accuracy of the LES boundary conditions a numerical experiment was performed which is similar to the numerical experiment by

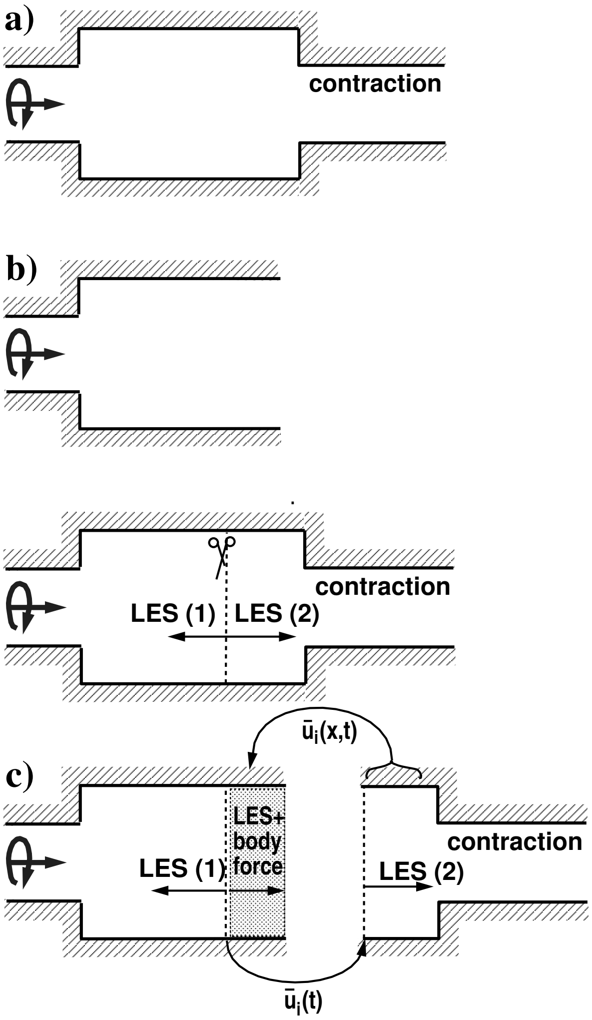

Fig. 2 Swirl Flow Geometry: a) full geometry, b) reduced LES domain, c) schematic splitting of domain to two computational domains

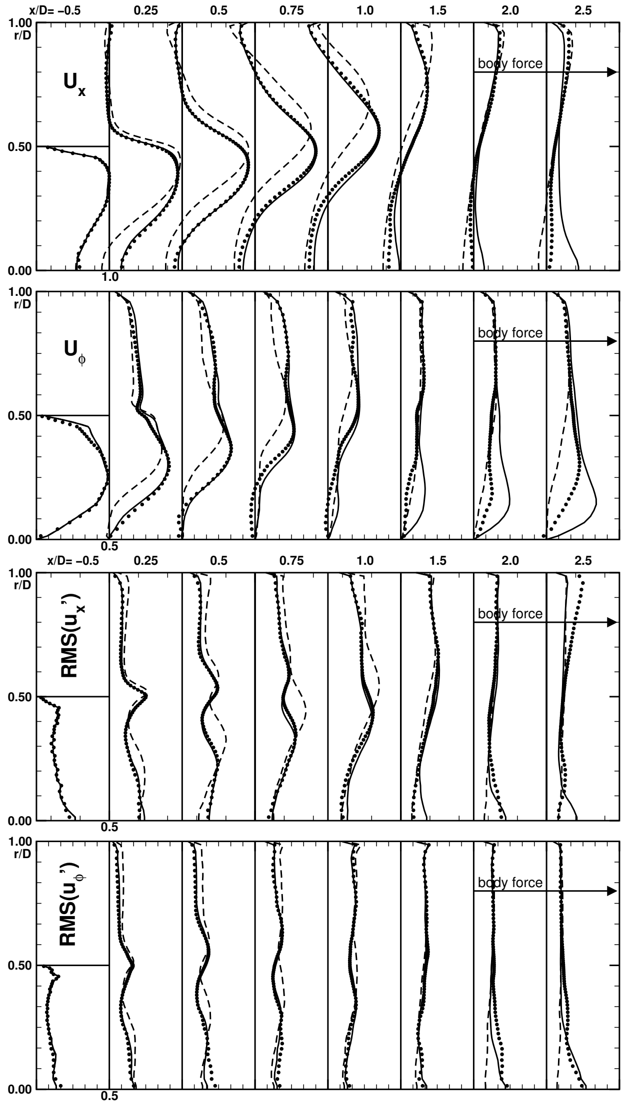

Fig. 3 Velocity components for different downstream positions. Solid line: LES domain a) (full geometry), dashed line: LES domain b) (reduced geometry), dots: integrated LES-LES computation c)

Schlüter *et al.*[7] to prove the applicability of the LES outflow conditions. A swirl flow at an expansion with a subsequent contraction three diameter D downstream of the expansion is considered (Fig. 2a). The swirl number of the flow is $S = 0.3$, which is just supercritical, meaning that vortex breakdown takes place and a recirculation zone develops. The extension and strength of this recirculation zone is strongly influenced by the presence of the downstream contraction. The simplicity of this test-case makes it possible to perform an LES computation of the complete geometry so that an 'exact' solution of the flow field is known (solid lines in Fig. 3).

Assuming, that in a final application, the usage of an LES flow solver is to be restricted to the computation of the expansion itself (Fig. 2b) it can be seen in Fig. 3 (dashed lines) that the solution differs substantially, if no correction is added in order to take the contraction downstream of the LES domain into account.

In a third computation, one instance of a LES code computes the expansion while another instance of the LES code computes the contraction (Fig. 2c). Both LES codes communicate to each other only by exchanging mean values at the interface, meaning that none of the two LES codes know, if the other code is a RANS or a LES flow solver. In the final application, the second LES flow solver is to be replaced by a RANS flow solver, but the usage of two LES codes excludes errors introduced by the different mathematical approaches of LES and RANS. It can be seen, that the simultaneous computation of the expansion and the contraction with the exchange of only the mean values delivers essentially the same result (black dots in Fig. 3) as the LES computation of the entire domain.

CONCLUSIONS

The integration of existing flow solvers to perform simultaneous computations can lead to an improvement in efficiency and accuracy of the solution due to the improved boundary treament. Here, it was possible to show that LES flow solvers are able to process solutions delivered by RANS flow solvers in order to obtain accurate results.

In the final version of this paper, results of actual integrated LES/RANS computations will be available and discussed. It is expected, that the RANS solution of the contraction differs from the LES solution, since RANS flow solvers usually have difficulties predicting highly swirling flows[3]. However, the flow prediction of the integrated solution of LES for the expansion and RANS for the contraction should be still better than the LES computation of the expansion alone. The computational cost of the integrated computation is to be expected only slightly higher than for the LES computation of the expansion (case b), but substantially lower than the computation of the expansion and the contraction with LES (case a).

The computation reported in this work proof the feasibility, accuracy and efficiency of integrated LES/RANS computations. This is an important step towards the application of this concept to realistic large scale flow systems such as the compressor/combustor, combustor/turbine or even complete compressor/combustor/turbine computations in order to predict multi-component effects such as compressor/combustor instability and combustor/turbine hotstreak migration.

ACKNOWLEDGMENTS

We gratefully acknowledge support by the US Department of Energy under the ASCI program.

References

[1] K. Akselvoll and P. Moin. Large-eddy simulation of turbulent confined coannular jets. *Journal of Fluid Mechanics*, 315:387–411, 1996.

[2] M. Germano, U. Piomelli, P. Moin, and W. Cabot. A dynamic subgrid-scale eddy viscosity model. *Phys. Fluids*, A(3 (7)):1760–1765, July 1991.

[3] B. Guo, T.A.G. Lnangrish, and D.F. Fletcher. Simulation of turbulent swirl flow in an axisymmetric sudden expansion. *AIAA Journal*, 41(1):96–102, 2001.

[4] P. Moin, K. Squires, W. Cabot, and S. Lee. A dynamic subgrid-scale model for compressible turbulence and scalar transport. *Phys. Fluids*, A 3(11):2746–2757, November 1991.

[5] C. Pierce and P. Moin. Large eddy simulation of a confined coaxial jet with swirl and heat release. *AIAA Paper*, (AIAA 98-2892), June 1998.

[6] C. D. Pierce and P. Moin. Method for generating equilibrium swirling inflow conditions. *AIAA Journal*, 36(7):1325–1327, 1998.

[7] J. U. Schluter, H. Pitsch, and P. Moin. Consistent boundary conditions for integrated les/rans simulations: Les outflow conditions. *AIAA paper*, (2002-3121), 2002. 32nd AIAA Fluid Dynamics Conference, June 24-27, St, Louis, MO.

[8] D. Veynante and T. Poinsot. *New Tools in Turbulence Modelling*, chapter 5 Reynolds averaged and large eddy simulation modeling for turbulent combustion, pages 105–140. Les edition physique. Springer, 1996.

[9] Yao, J., Jameson, A., Alonso, J. J., and Liu, F., Development and Validation of a Massively Parallel Flow Solver for Turbomachinery Flows. AIAA Paper 00-0882, Reno, NV, January 2000.

[10] Jameson, A., Time Dependent Calculations Using Multigrid, with Applications to Unsteady Flows Past Airfoils and Wings. AIAA Paper 91-1596, AIAA 10th Computational Fluid Dynamics Conference, Honolulu, HI, June 1991.

[11] Alonso, J. J., Martinelli, L., and Jameson, A., Multigrid Unsteady Navier-Stokes Calculations with Aeroelastic Applications. AIAA Paper 95-0048, AIAA 33rd Aerospace Sciences Meeting and Exhibit, Reno, NV, 1995.

[12] Belov, A., Martinelli, L., and Jameson, A., Three-Dimensional Computations of Time-Dependent Incompressible Flows with an Implicit Multigrid-Driven Algorithm on Parallel Computers. Proceedings of the 15th International Conference on Numerical Methods in Fluid Dynamics, Monterey, CA, June 1996.