AIAA 2004-0369

Integrated RANS-LES Computations in Gas Turbines: Compressor-Diffusor Coupling

J. U. Schlüter, X. Wu, S. Kim, J. J. Alonso and H. Pitsch

Center for Turbulence Research &

Aerospace Computing Lab

Stanford University, Stanford, CA

42nd Aerospace Sciences Meeting and Exhibit Conference

January 5–9, 2004/Reno, NV

INTEGRATED RANS-LES COMPUTATIONS IN GAS TURBINES: COMPRESSOR-DIFFUSOR COUPLING

J. U. Schlüter,* X. Wu,† S. Kim,‡ J. J. Alonso§ and H. Pitsch¶

Center for Turbulence Research &
Aerospace Computing Lab
Stanford University, Stanford, CA

Multi-component effects, such as compressor-combustor interactions, are difficult to predict with computational fluid dynamics using reasonable computational resources due to the variety of physical phenomena, which have to be modeled. The approach described in the present study is to couple two separate flow solvers, a RANS flow solver computing the last stage of a compressor and an LES flow solver computing the prediffuser of a combustor. At the interface the flow solvers exchange the flow data in order to define each others boundary conditions at the interface plane. The use of multiple flow solvers allows the application of the most appropriate flow solver for each flow section. The present study describes the approach in general, the interface, which allows running multiple flow solvers, its validation, and presents its application to the compressor-diffuser example.

INTRODUCTION

IN the design process of gas turbine engines computational fluid dynamics (CFD) is usually used to predict the flow in single components of the engine, such as the compressor, the combustor, or the turbine. The simulation of the *entire* flow path of a gas turbine engine using today's flow solvers is prohibited by the enormous computational costs. However, the increasing computational resources and the improved efficiency of future flow solvers puts the simulation of an entire engine within reach. In order for such a simulation to be useful in the design process it has to deliver accurate results within a few weeks.

The goal of the Accelerated Strategic Computing Initiative (ASCI) of the Department of Energy (DoE) at Stanford is to develop high performance flow solvers, which are able to use highly parallel super-computers, for the simulation of an entire engine. While the development of new super-computers is one of the main tasks in the overall ASCI effort of the DoE, the ASCI project at Stanford investigates the development of flow solvers for gas turbine engines in order to improve efficiency, scalability, and modeling of physical effects. However, looking at the wide variety of the flow phenomena, which have to be simulated in the flow path of the engine, it can be seen that only the use of multiple specialized flow solvers, one for the turbo-machinery parts and one for the combustor, can guarantee the efficiency and accuracy of a simulation. The reason for that is, that the flow regimes and the physical effects, which have to be modeled vary dramatically in these two components. Most flow solvers used nowadays in the design process are specialized for one of the two tasks.

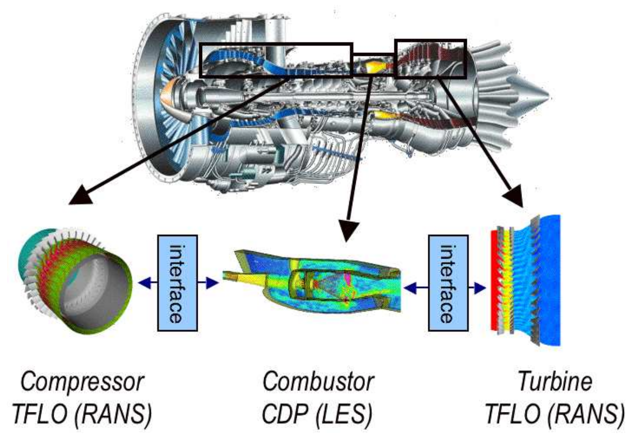

Fig. 1 Computation of the flow path of an entire gas turbine: Decomposition of the engine. Compressor and turbine with RANS; Combustor with LES (Additional images from[1,2])

The flow field in the turbomachinery parts is characterized by high Reynolds-numbers and high Mach-numbers. The accurate prediction of the flow requires the precise description of the turbulent boundary layers around the rotor and stator blades. A variety of flow solvers have been developed to deal with these flow configurations and are in use in industry since many years. These flow solvers are based on the Reynolds-averaged Navier-Stokes (RANS) approach. Here, the mean flow field is computed delivering an ensemble-averaged solution. All turbulent motions are modeled with a turbulence model. Due to the complexity of the flow in the turbo-machinery, the turbulence models have to be adapted with some of its parameters in order to deliver an accurate solution. Since this kind of flow is subject of investigation since

*Research Associate, Member AIAA
†Research Associate
‡Postdoc
§Assistant Professor, Member AIAA
¶Associate Professor

many years, these parameters are usually well known and hence, the flow solvers deliver reasonably good results.

The flow in the combustor on the other hand is characterized by detached flows, chemical reactions and heat release. The prediction of detached flows and free turbulence is enhanced using flow solvers based on Large-Eddy Simulations (LES). And while the use of LES increases the computational costs, LES has been the only predictive tool which is able to simulate consistently these complex flows. LES resolves the large scale turbulent motions in time and space and models only the smallest scales, which are usually more universal and hence, easier to model[3] . Since most of the turbulent scales are resolved, the modeling of the flame is facilitated by the additional data that is available due to the temporal resolution[4] . LES flow solvers have shown in the past that they are able to model simple flames and are more and more adapted for use in gas turbine combustors[2,5] .

In order to predict multi-component effects, such as compressor-combustor instability, combustor-turbine hot-streak migration and combustion instabilities, these flow solvers have to run simultaneously, each computing its part of the domain, and exchange flow information at the interface (Fig. 1). The parallel execution of multiple flow solvers requires the definition of an interface, which allows the exchange of flow information and a framework of boundary conditions in order to process the exchanged data.

The current study describes the flow solvers and the interface used in this project. The approach of multiple flow solvers is then validated on a simple test-case. Furthermore, in order to demonstrate the feasibility of coupled RANS-LES computations in turbomachinery applications, the last stage of a compressor computed with RANS is coupled with an LES of the prediffuser of the combustor. This flow configuration is important, since the outflow of the compressor alters the flow field in the subsequent diffuser[6,7] . A detailed knowledge of the flow field in this section allows to optimize the diffuser design in order to to achieve a decrease of pressure loss. Here, the NASA stage 35 compressor is coupled with a generic diffuser in order to assess the integrated RANS-LES approach and to test coupled RANS-LES for the upstream interface for a computation of an entire engine (Fig. 1).

RANS FLOW SOLVER

RANS flow solvers are solving the classical Reynolds-Averaged Navier-Stokes equations for turbulent flows. Here, the flow variables are split into a mean and a fluctuating part $u_i = \bar{u}_i + u'_i$ and the Navier-Stokes equations are time-averaged. This delivers a set of equations for the mean velocities, but leaves an unclosed term $\overline{u'_i u'_j}$, which has to be modeled with a turbulence model. Turbulence models are commonly based on the eddy viscosity approach, where the eddy viscosity can be modeled in varying levels of complexity. The most applied models for RANS flow solvers are two-equation models, such as the $k-\epsilon$ or $k-\omega$ models, where two additional transport equations are solved in order to determine the eddy viscosity. These models are accepted as a good compromise for turbo-machinery applications between efficiency and accuracy.

The RANS flow solver used for this investigation is the TFLO code developed at the Aerospace Computing Lab (ACL) at Stanford. The flow solver computes the unsteady Reynolds Averaged Navier-Stokes equations using a cell-centered discretization on arbitrary multi-block meshes[8] . The solution procedure is based on efficient explicit modified Runge-Kutta methods with several convergence acceleration techniques such as multi-grid, residual averaging, and local time-stepping. These techniques, multi-grid 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[9–11] is used for time-accurate simulations that account for the relative motion of moving parts as well as other sources of flow unsteadiness.

LES FLOW SOLVER

LES flow solvers solve for the filtered Navier-Stokes equations. The filter ensures that the large scale turbulence is resolved in time and space resulting in a decomposition of the variables in a resolved and a subgrid part $u_i = \tilde{u}_i + u_i"$. For practical purposes, usually the mesh filter is applied, which means that the cell size defines the filter at each location. Applying the filter to the Navier-Stokes equation leaves an unclosed term $\widetilde{u_i" u_j"}$, which defines the subgrid turbulence. As opposed to the similar unclosed term $\overline{u'_i u'_j}$ from the RANS flow solver, which includes the turbulent motions of all scales, the LES term describes only the subgrid turbulence. With a sufficiently high mesh resolution, the LES solution is rather robust against the chosen subgrid model. Most models use an eddy viscosity approach to model the subgrid stresses. Here, the eddy viscosity can be determined by algebraic models such as the Standard Smagorinsky model,[12] or, as used in this study, by a dynamic procedure, where the solution of the high frequent resolved flow field is used to determine the subgrid stresses[13] .

The LES flow solver used for the current study is the CDP-α code developed at the Center for Turbulence Research (CTR) at Stanford. The filtered momentum equations are solved on a cell-centered unstructured mesh and are second-order accurate. An implicit time-advancement is applied. The subgrid stresses are modeled with a dynamic procedure.

INTERFACE

Part of the efforts to integrate these flow solvers is the definition of the interface. The optimization of the communication and the processing of the exchanged data to meaningful boundary conditions are some of the encountered challenges. In previous work interface routines have been established and validated with simple geometries[14–16] .

The interface used for establishing a connection between the flow solvers consists of routines following an identical algorithm in all flow solvers. The message passing interface MPI is used to create communicators, which are used to communicate data directly between the individual processors of the different flow solvers. This means that each processor of one flow solver can communicate directly with all of the processors of the other flow solvers. This requires the interface routines to be part of the source code of all flow solvers. A detailed description of the common algorithms can be found in Schlüter *et al.*[17,18] .

In a handshake routine, each processor determines whether its domain contains points on the interface. The location of these points are sent to all processors of the other peer flow solvers. The processors of the peer flow solvers then determine and communicate back, whether the received points are within their own domain. During the actual flow computation all processors communicate data for a common point directly with each other.

The approach of embedding the interface into the source code of each flow solver has been chosen for its efficiency in the communication process. Alternative solutions would be to use a third code, which organizes the communication between the flow solvers, or to limit the peer-to-peer communication to the root processes of each flow solver. While the latter two solutions are usually easier to implement, they cause more communication processes and slow down the computation.

BOUNDARY CONDITIONS

The definition of the boundary conditions requires special attention especially on the LES side due to the different mathematical approaches. Since on the LES side part of the turbulent spectrum is resolved, the challenge is to regenerate and preserve the turbulence at the boundaries. At the LES outflow, a body force method has been developed to impose RANS solutions at the outflow of the LES domain[19,20] .

At the LES inflow boundary, the challenge is to prescribe transient turbulent velocity profiles from ensemble-averaged RANS data. Simply adding random fluctuations to the RANS profiles miss the temporal and spatial correlations of real turbulence and are dissipated very quickly. Instead, a data-base of turbulent fluctuations is created by an auxiliary LES computation of a periodic turbulent pipe flow. The LES inflow boundary condition can then be described as[17,18,21] :

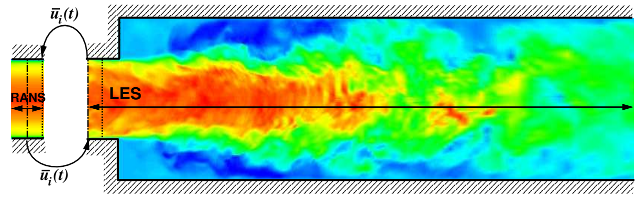

Fig. 2 Interface validation: Integrated RANS-LES of a confined jet.

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

with the sub-script RANS denoting the solution obtained from the RANS computation and quantities with sub-script DB are from the database. Here, t is the time, u_i stands for the velocity components, and $\overline{u}_i$ is the ensemble average of the velocity component u_i.

Term II of Eq. (1) is the velocity fluctuation of the database. This turbulent fluctuation is scaled to the desired value by multiplication with term III, which ensures that the correct level of velocity fluctuation is recovered.

On the RANS side, Navier-Stokes Characteristic Boundary Conditions (NSCBC)[22] are applied using the time-averaged solution from the LES side.

RANS-LES INTERFACE VALIDATION

While most of the fundamental issues of integrated RANS-LES computations of previous studies were addressed using a structured LES flow solver in order to decrease computational costs, the envisioned increase in complexity of the geometries calls for the use of an unstructured flow solver. Hence, the described interface was implemented into the unstructured LES flow solver CDP-α.

As a validation of the interface and the LES inflow boundary condition, a coupled RANS-LES computation of an axisymmetric expansion has been performed. The test-case corresponds to the experimental configuration of Dellenback *et al.* (1988). Here, a part of the flow domain upstream of the expansion is computed with a RANS code (Fig. 2).

The inlet velocity profiles in the RANS section are specified according to the experimental data at this location. The RANS flow solver TFLO computes the flow through the upstream pipe and at its outlet hands over the data to the subsequent LES flow solver. The RANS domain is relatively short ($0.5D$, with D being the diameter of the pipe upstream of the expansion.)

The LES flow solver CDP obtains its inflow velocity profiles from the RANS flow solver and specifies its

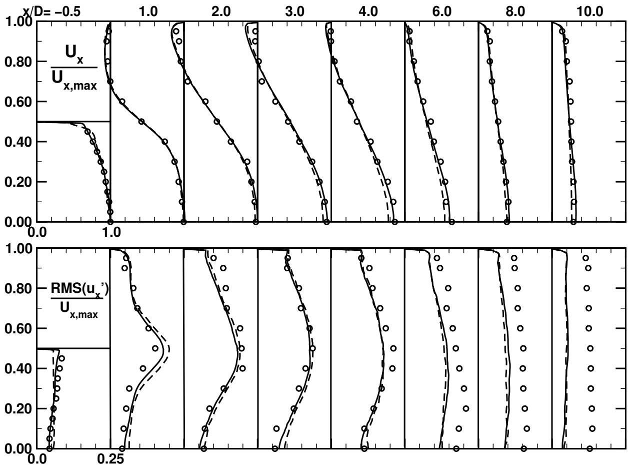

Fig. 3 Results of interface validation. Above: axial velocity profiles. Below: Axial velocity fluctuations. Circles: experiments. Solid lines: LES alone defining inflow from experimental data. Dashed lines: Integrated RANS-LES, RANS defining inflow from experimental data, LES defining inflow from simultaneously running RANS.

LES inflow boundary conditions according to Eq. 1.

The results of the integrated computation are then validated against the experimental data and verified against an LES computation using an inflow data-base at the inlet in which the data-base statistics are corresponding to the experimental data at the inlet plane.

The RANS mesh contains 350,000 mesh points and is refined near the wall. The LES mesh contains 1.1 million mesh points with the mesh points concentrated near the spreading region of the jet. The far field of the jet is relatively coarse.

Fig. 3 shows the LES velocity profiles obtained from this computation. The integrated TFLO-CDP computation predicts essentially the same results as the single LES computation and matches the experimental data well. Please note that the far field of the jet is not well resolved and hence, the turbulent fluctuations in the far field are underestimated by both LES computations.

INTEGRATED RANS-LES OF A COMPRESSOR-PREDIFFUSER

In order to test the applicability of coupled RANS-LES computations in realistic geometries, a turbomachinery case has been investigated. The goal of this study is to test the interface routines for the flow between the compressor and the combustor and to study the influence of possible unsteady interactions of compressor and the combustor inlet diffuser. The test-case consists of a compressor geometry computed by a RANS flow solver and a pre-diffuser, which is a component upstream of the injector to the combustor, computed by an LES flow solver.

The computational study of such cases is relevant and important, since typically these two components are developed in isolation and combined tests are done only in the final prototype assembly. The numerical prediction of this flow configuration would allow to assess the interactions of the components during the design phase of the engine. One of the biggest questions in compressor-prediffuser flows is whether separation in the diffuser takes place. Since the inflow of the pre-diffuser is inhomogeneous and periodically perturbed by blade passings, the integrated computation of this geometry can offer insights on how to modify the geometry in order to develop a more compact, non-separating diffuser.

The drawback of the choice of this configuration is that no experimental data exists to validate the computation. The quality of the computed results can only be guaranteed on the basis of the separate validation process that the component codes have undergone and the detailed testing of the interface routines that has been presented in previous work. Some validation studies of the individual flow solvers are given in Yao *et al.*[8] and Davis *et al.*[1,23] for the TFLO code and in Mahesh *et al.*[24] and Constantinescu *et al.*[2] for the CDP code. The interface has been developed and tested in detail over the last two years[14–16] . While many of the techniques necessary for coupling these two flow solvers are still under development, all necessary elements, such as the coupling procedure and the boundary conditions on both sides are currently in place for the chosen test-case.

The goal of this computation is to demonstrate the feasibility of integrated RANS-LES computations in a turbomachinery environment and to identify practical issues involved in these calculations.

Geometry

The compressor geometry for the computed test-case corresponds to that of a modified NASA experimental rig stage 35. The experimental rig consists of a row of 46 rotors and a row of 36 stators. In order to simplify this geometry, the rotor stage has been rescaled to a 36 blade count, which allows to compute an axisymmetric segment of 10° using periodic boundary conditions at the corresponding azimuthal planes.

For this integrated computation, the rotor tip-gap has been closed in order to decrease the overall computational costs. The inclusion of the tip-gap is addressed in the TFLO flow solver and poses no additional problem from the integration point of view. The RANS time step was chosen to resolve one blade passing with 50 intervals.

The RANS mesh is a structured multi-block mesh consisting of approximately 1.5 million control volumes. The speed of the rotor was set to a relatively low 5000 RPM in order to keep the flow at the interface within the low-Mach number regime that the LES solver is able to handle. This decrease in rotational speed had to be done for the current case. In a real

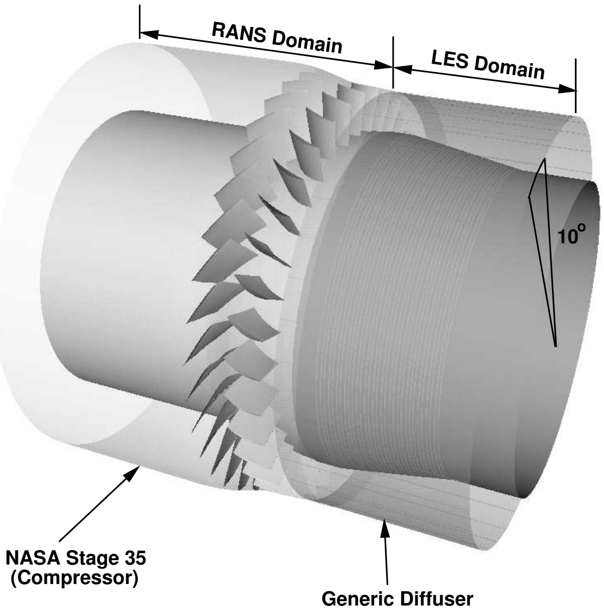

Fig. 4 Geometry of coupled NASA stage 35/prediffuser. RANS domain includes one rotor and one stator. LES domain includes the diffuser. A 10° axisymmetric sector is computed.

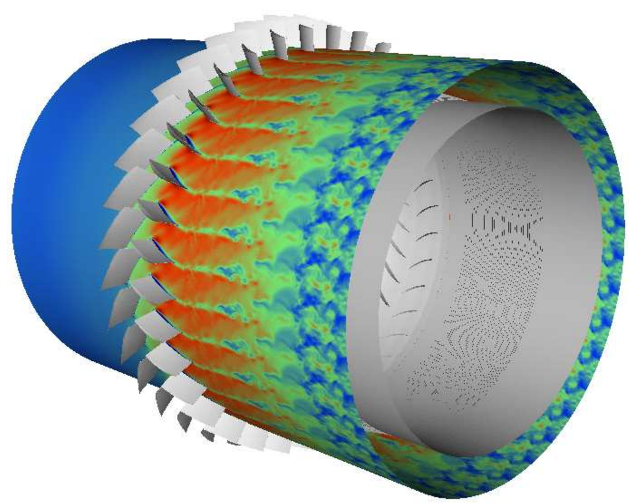

Fig. 5 Integrated RANS-LES of compressor/prediffuser: Velocity distribution at the 50% plane.

engine, the compressor consists of a multiple stages resulting in a higher pressure and a higher temperature at the compressor exit. The high temperature of the air in this section of the flow path will ensure that the low-Mach number approximation is not violated, even when the engine is at full load.

For the RANS domain, the flow solver TFLO has been used. On the LES side, the LES flow solver CDP-α has been applied.

The diffuser expands one stator chord length behind the stator. The LES domain starts 1/3 chord behind the stator. The RANS domain reaches 2/3 of the chord length into the LES domain, which essentially means that the RANS outlet plane is just at the expansion of the diffuser.

The diffuser geometry has been chosen with a relatively wide opening such that separation may occur. The diffuser opens towards the centerline of the compressor. Over 3 chord lengths, the diffuser opens up 0.5 chord lengths. The outer wall of the diffuser is straight.

The LES mesh for the CDP flow solver consists of 500,000 control volumes and is concentrated near the walls. LES inflow boundary conditions were defined corresponding to Eq. 1.

In order to initialize the solutions in both domains, separate computations were performed. On the basis of the initial, separate computations, the computational needs for each domain and solver were assessed in order to balance the split of processors for the computation. The load balancing between the two flow solvers has to be done manually, since the current version of MPI does not support a dynamic splitting of the processors using multiple codes.

Results

The computations using the unstructured LES flow solver CDP-α and TFLO was carried out using 64 processors for TFLO and 64 processors for CDP-α. Here, 8 blade passings were computed in 60 hours of wall clock time using an IBM Power3.

The actual Mach number at the interface was Ma= 0.1 ensuring the validity of the low-Mach number approximation in the LES domain. The mass flux over the interface was conserved with an error of $\approx 0.5\%$.

Figures 5 and 6 show the axial velocity distributions at 50% span of the compressor blades for an instantaneous snapshot of the computation. The upstream RANS solution corresponds to a phase averaged solution while the downstream LES solution is truly unsteady.

The wakes of the stators can clearly be identified in the RANS domain downstream of the stators. The communication of the flow solvers at the interface ensures that the full 3D flow features are transferred from the upstream flow solver to the downstream domain. The boundary conditions of the LES flow solver are defined according to these data. Hence, the wake of the stator correctly propagates across the interface and can still be found far downstream in the diffuser. It can also be seen that the turbulence, which is resolved in the LES domain, creates a more disturbed velocity distribution.

The differences in the description of turbulence are more apparent in Fig. 7, which shows the vorticity distribution at 50% span of the stator. Here the magnitude of the vorticity is depicted computed according to the unsteady flow field of both domains. In the RANS domain, the vorticity is mainly created due to the mean flow features, such as wall boundary layers, and secondary flows and vortices. The stator creates

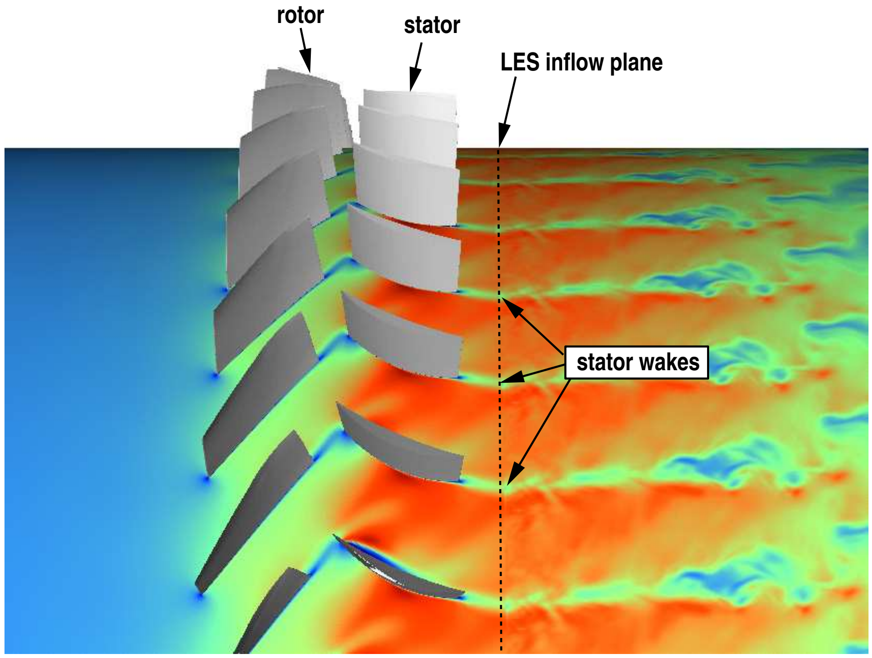

Fig. 6 Integrated RANS-LES of compressor/prediffuser: Velocity distribution at the 50% plane. Close-up of the interface.

two vorticity sheets, one on the extrado, one on the intrado. Both vorticity sheets propagate downstream across the interface.

The vorticity distribution in the LES domain is characterized by small scale turbulence. Turbulence present in the upstream RANS domain and modeled by a RANS turbulence model has to be regenerated. The small scale turbulence has been reconstructed at the interface using the LES inflow boundary condition (Eq. 1.) It can be seen that the small-scale turbulence interferes with the stator wakes. The turbulent diffusion of the stator wakes in the RANS domain is modeled with an eddy viscosity model, which gives them a very smooth appearance. In the LES domain, the turbulent transport is given by the resolved turbulence, and hence, vortical turbulent structures can be identified.

Time data recorded on the LES side of the interface did not reveal a predominant frequency such as the blade passing frequency. This could be due to the relatively short computed time-span of eight blade passings or due to the low rotational speed of the compressor. Future investigations will study the presence of predominant frequencies in the flow due to blade passings in more detail.

CONCLUSIONS

The approach for coupled RANS-LES computations in order to improve accuracy and efficiency of turbomachinery-combustor computations has been presented. The computation of the coupled NASA stage 35/prediffuser geometry demonstrated the concept of integrated RANS-LES computations in a realistic environment.

Future work will focus on real aircraft engine geometries in order to characterize the flow in the prediffuser. Furthermore, computations for the prediction of the entire flow path of a gas turbine will be prepared.

ACKNOWLEDGMENTS

The support by the US Department of Energy under the ASCI program is gratefully acknowledged.

References

[1]Davis, R., Yao, J., Clark, J. P., Stetson, G., Alonso, J. J., Jameson, A., Haldeman, C., and Dunn, M., 2002. "Unsteady interaction between a transsonic turbine stage and downstream components". ASME Turbo Expo 2002 (GT-2002-30364) [].

[2]Constantinescu, G., Mahesh, K., Apte, S., Iaccarino, G., Ham, F., and Moin, P., 2003. "A new paradigm for simulation of turbulent combustion in realistic gas turbine combustors using LES". ASME Turbo Expo 2003 (GT2003-38356) [].

[3]Ferziger, J. H., 1996. *New Tools in Turbulence Modelling*, vol. New Tools in Turbulence Modelling of *Les edition physique*. Springer, chapter 2 Large eddy simulation: an introduction and perspective, pp. 29–47.

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

[5]Poinsot, T., Schlüter, J., Lartigue, G., Selle, L., Krebs, W., and Hoffmann, S., 2001. "Using large eddy simulations to understand combustion instabilities in gas turbines". *IUTAM Symposium on Turbulent Mixing and Combustion* [], pp. 1–8. Kingston, Canada 2001.

[6]Barker, A. G., and Carrotte, J. F., 2001. "Influence of compressor exit conditions on combustor annular diffusers, part 1: Diffuser performance". Journal of Propulsion and Power, **17** (3) [], pp. 678–686.

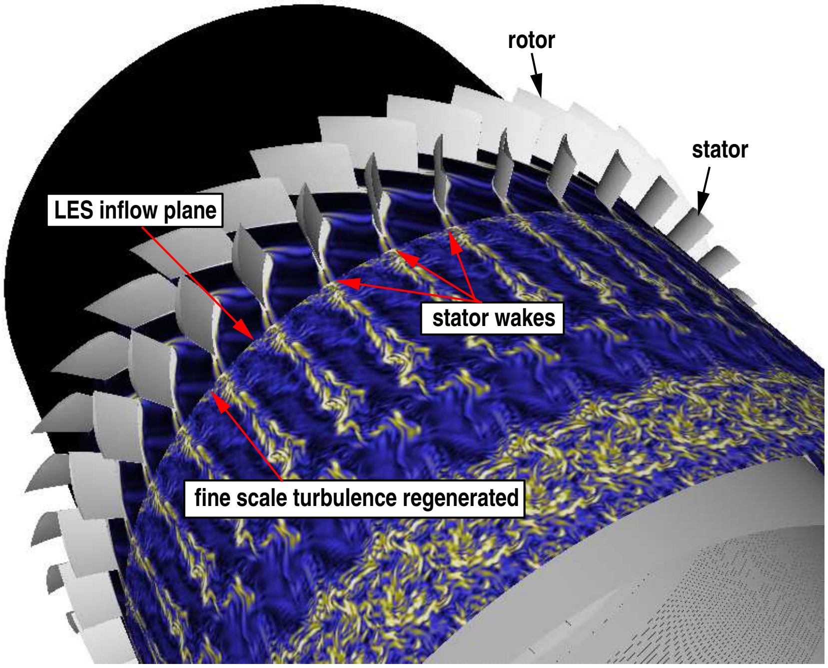

Fig. 7 Integrated RANS-LES of compressor/prediffuser: Vorticity magnitude distribution at the 50% plane. Vorticity created on the surfaces of the stators can be found in the LES domain.

[7]Barker, A. G., and Carrotte, J. F., 2001. "Influence of compressor exit conditions on combustor annular diffusers, part 2: Flow redistribution". Journal of Propulsion and Power, **17** (3) [], pp. 687–694.

[8]Yao, J., Jameson, A., Alonso, J. J., and Liu, F., 2000. "Development and validation of a massively parallel flow solver for turbomachinery flows". AIAA paper (AIAA-00-0882) [].

[9]Jameson, A., 1991. "Time dependent calculations using multigrid, with applications to unsteady flows past airfoils and wings". AIAA paper (AIAA Paper 91-1596) []. AIAA 10th Computational Fluid Dynamics Conference, Honolulu, HI, June 1991.

[10]Alonso, J. J., Martinelli, L., and Jameson, A., 1995. "Multigrid unsteady Navier-Stokes calculations with aeroelastic applications". AIAA Paper (AIAA 95-0048) []. AIAA 33rd Aerospace Sciences Meeting and Exhibit, Reno, NV, 1995.

[11]Belov, A., Martinelli, L., and Jameson, A., 1996. "Three-dimensional computations of time-dependent incompressible flows with an implicit multigrid-driven algorithm on parallel computers". In Proceedings of the 15th International Conference on Numerical Methods in Fluid Dynamics, Monterey, CA.

[12]Smagorinsky, J., 1963. "General circulation experiments with the primitive equations, I, the basic experiment". Mon. Weather Rev., **91** (3) [], pp. 99–152.

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

[14]Shankaran, S., Liou, M.-F., Liu, N.-S., and anf J. J. Alonso, R. D., 2001. "A multi-code-coupling interface for combustor/turbomachinery simulations". AIAA paper (AIAA 2001-0974) [].

[15]Schlüter, J. U., Shankaran, S., Kim, S., Pitsch, H., Alonso, J. J., and Moin, P., 2003. "Integration of RANS and LES flow solvers for simultaneous flow computations". AIAA paper (AIAA 2003-0085) [].

[16]Schlüter, J. U., Shankaran, S., Kim, S., Pitsch, H., Alonso, J. J., and Moin, P., 2003. "Towards multi-component analysis of gas turbines by CFD: integration of RANS and LES flow solvers". ASME paper (ASME GT2003-38350) []. ASME Turbo Expo 2003, June 16-19, 2003, Atlanta, GA.

[17]Schlüter, J. U., Pitsch, H., and Moin, P., 2003. "Boundary conditions for LES in coupled simulations". AIAA paper (AIAA-2003-0069) [].

[18]Schlüter, J. U., Pitsch, H., and Moin, P., 2003. "LES inflow boundary conditions for coupled RANS-LES computations". *accepted, AIAA Journal* [].

[19]Schlüter, J. U., Pitsch, H., and Moin, P., 2002. "Consistent boundary conditions for integrated LES/RANS simulations: LES outflow conditions". AIAA paper (2002-3121) []. 32nd AIAA Fluid Dynamics Conference, June 24-27, St, Louis, MO.

[20]Schlüter, J. U., Pitsch, H., and Moin, P., 2003. "LES outflow conditions for integrated LES/RANS simulations". *submitted for publication to AIAA Journal* [].

[21]Schlüter, J. U., 2003. "Consistent boundary conditions for integrated LES/RANS computations: LES inflow conditions". AIAA paper (AIAA-2003-3971) []. 16th AIAA CFD conference 2003.

[22]Poinsot, T. J., and Lele, S. K., 1992. "Boundary conditions for direct simulations of compressible viscous reacting flows". Journal of Computational Physics (101) [], pp. 104–129.

[23]Davis, R., Yao, J., Alonso, J. J., Paolillo, R., and Sharma, O. P., 2003. "Prediction of main/secondary-air system flow interaction in a high pressure turbine.". AIAA Paper (AIAA-2003-4833) [].

[24]Mahesh, K., Constantinescu, G., Apte, S., Iaccarino, G., and Moin, P., 2001. "Large-eddy simulations of gas turbine combustors". *Annual Research Briefs 2001* [], pp. 3–18. Center for Turbulence Research, NASA Ames/Stanford Univ.