Integrated RANS-LES computations of turbomachinery components: Generic compressor/diffuser

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

1. Introduction

The goal of the ASCI project at Stanford is the computation of the entire aero-thermal flow in an aircraft gas turbine engine. As part of the project, high performance flow solvers are developed to address the prediction of the flow in components of the turbine. For the turbomachinery parts, a flow solver based on Reynolds-Averaged Navier-Stokes (RANS) approach is used. This flow solver is validated on a variety of turbomachinery applications (Davis *et al.* 2002; Davis *et al.* 2003) and is scalable to run on a large number of processors in parallel, a feature necessary in light of the enormous task of the final application. For the prediction of the reacting flow in the combustion chamber, a flow solver based on Large Eddy Simulations (LES) is used. The strongly detached flow in the combustor requires that the numerical approach has to resolve the large scale turbulence in time and space in order to predict the flow features accurately. Furthermore, the temporal resolution of the flow is very beneficial for the modeling of the reactive flow. This makes LES much more suitable for this portion of the flow path. An LES flow solver capable of modeling the variety of physical phenomena, such as turbulence, spray and heat release, is currently under development and in the process of validation (Mahesh *et al.* 2001; Constantinescu *et al.* 2003).

In order to predict multi-component phenomena, such as compressor-combustor instability, combustor-turbine hot-streak migration and combustion instabilities, these RANS and LES flow solvers have to run simultaneously, each computing its part of the gas turbine. At the interfaces of the individual domains, the flow solvers have to communicate the flow parameters required to evaluate appropriately defined boundary conditions.

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 (Shankaran *et al.* 2001; Schlüter *et al.* 2003d; Schlüter *et al.* 2003e). Since the definition of boundary conditions on the LES side requires special attention, LES inflow (Schlüter *et al.* 2003a; Schlüter, 2003; Schlüter *et al.* 2003b) and outflow boundary conditions (Schlüter *et al.* 2002; Schlüter *et al.* 2003c) have been established. While most of this work has been carried out on simple test-cases, the value of coupled RANS-LES computation can only be assessed in the application to industrial problems. The next logical step is to apply the coupled RANS-LES approach to complex geometries of turbomachinery applications.

The goal of the current study is to apply the coupled RANS-LES approach to a compressor-diffuser geometry of a gas turbine. This flow configuration is important, since the outflow of the compressor alters the flow field in the subsequent diffuser (Barker & Carrotte 2001a; Barker & Carrotte 2001b). A detailed knowledge of the flow field in this

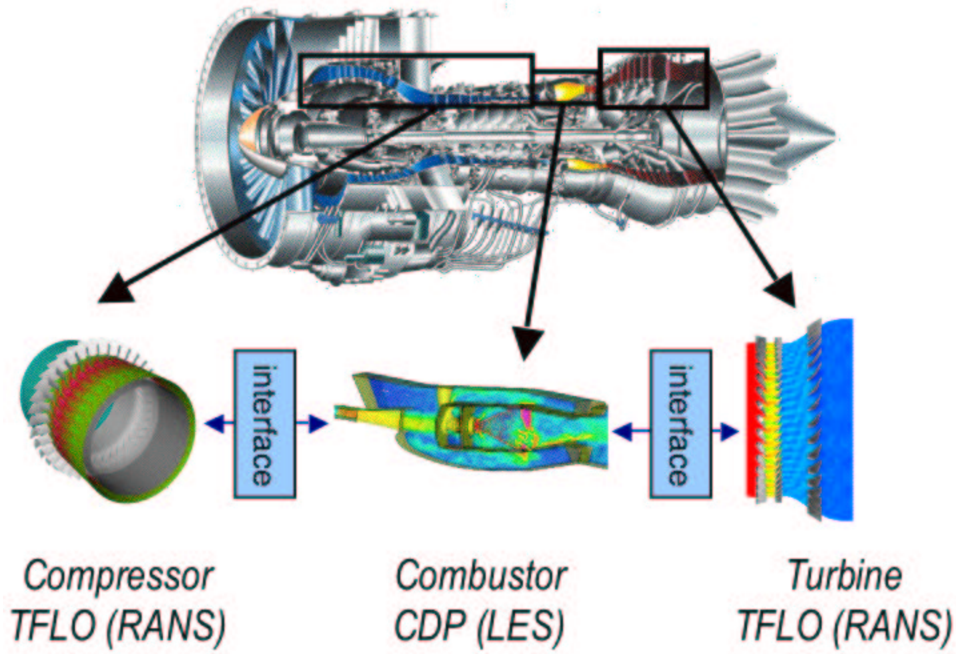

FIGURE 1. Decomposition of gas turbine engine. LES of combustor(Mahesh *et al.* 2001; Constantinescu *et al.* 2003) , RANS of turbine section (Davis *et al.* 2002

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.

2. Interface Description

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.* (2003d and 2003e).

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

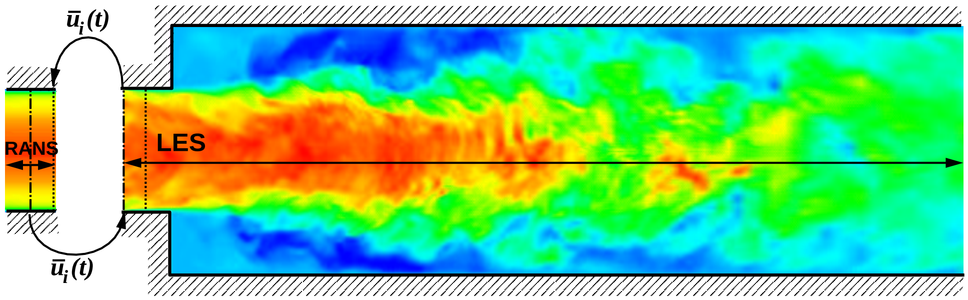

FIGURE 2. Interface validation for CDP-α: Integrated RANS-LES computation of a confined jet.

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.

The interface routines are written in a modular approach, where the communication steps are separated from code specific routines. This facilitates the implementation of the interface into other flow solvers.

3. CDP-α Interface Validation

While most of the fundamental issues of integrated RANS-LES computations in previous work 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.

CDP-α has been developed to predict chemically reacting flows in gas turbine combustors. The filtered momentum equations based on a low-Mach number approximation are solved with a 2nd order implicit time-advancement. A dynamic model is used for the turbulent sub-grid stresses.

LES inflow boundary conditions as proposed by Schlüter *et al.* (2003b) have been implemented into CDP-α. Here, 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:

$$u_{i,\mathrm{LES}}(t) = \underbrace{\overline{u}_{i,\mathrm{RANS}}(t)}_{I} + \underbrace{(u_{i,\mathrm{DB}}(t) - \overline{u}_{i,\mathrm{DB}})}_{II} \cdot \underbrace{\frac{\sqrt{\overline{u'^2_{(i)}}}_{\mathrm{RANS}}(t)}{\sqrt{\overline{u'^2_{(i)}}}_{\mathrm{DB}}}}_{III} \tag{3.1}$$

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. (3.1) is the velocity fluctuation of the database. This turbulent fluctu-

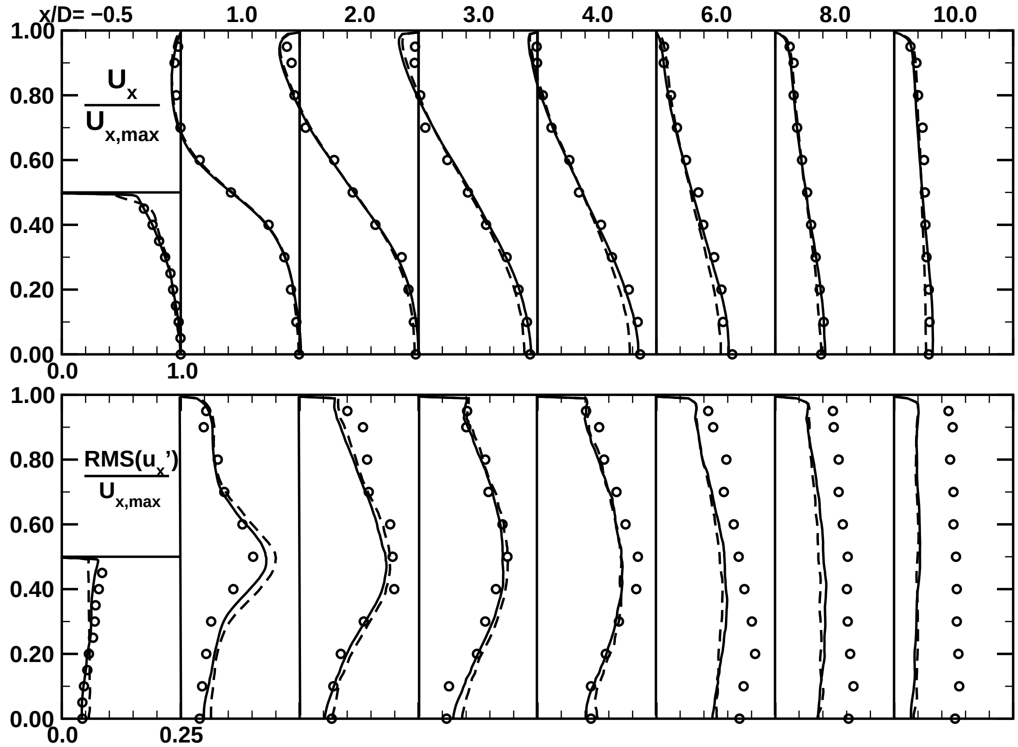

FIGURE 3. Velocity profiles of computation of a confined jet. Circles: experiment, solid line: CDP-α defining inlet from experiments, dashed line: integrated TFLO-CDP computation.

ation is scaled to the desired value by multiplication with term III, which ensures that the correct level of velocity fluctuation is recovered.

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 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 (Yao *et al.* 2000). 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 (Jameson 1991; Alonso *et al.* 1995; Belov *et al.* 1996) is used for time-accurate simulations that account for the relative motion of moving parts as well as other sources of flow unsteadiness.

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 LES inflow boundary conditions according to Eq. 3.1.

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.

4. Integrated RANS-LES of a Realistic Turbomachinery Geometry

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.* (2000) and Davis *et al.* (2002 and 2003) for the TFLO code and in Mahesh *et al.* (2001) and Constantinescu *et al.* (2003) for the CDP code. The interface has been developed and tested in detail over the last two years (Shankaran *et al.* 2001; Schlüter *et al.* 2003d; Schlüter *et al.* 2003e). 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

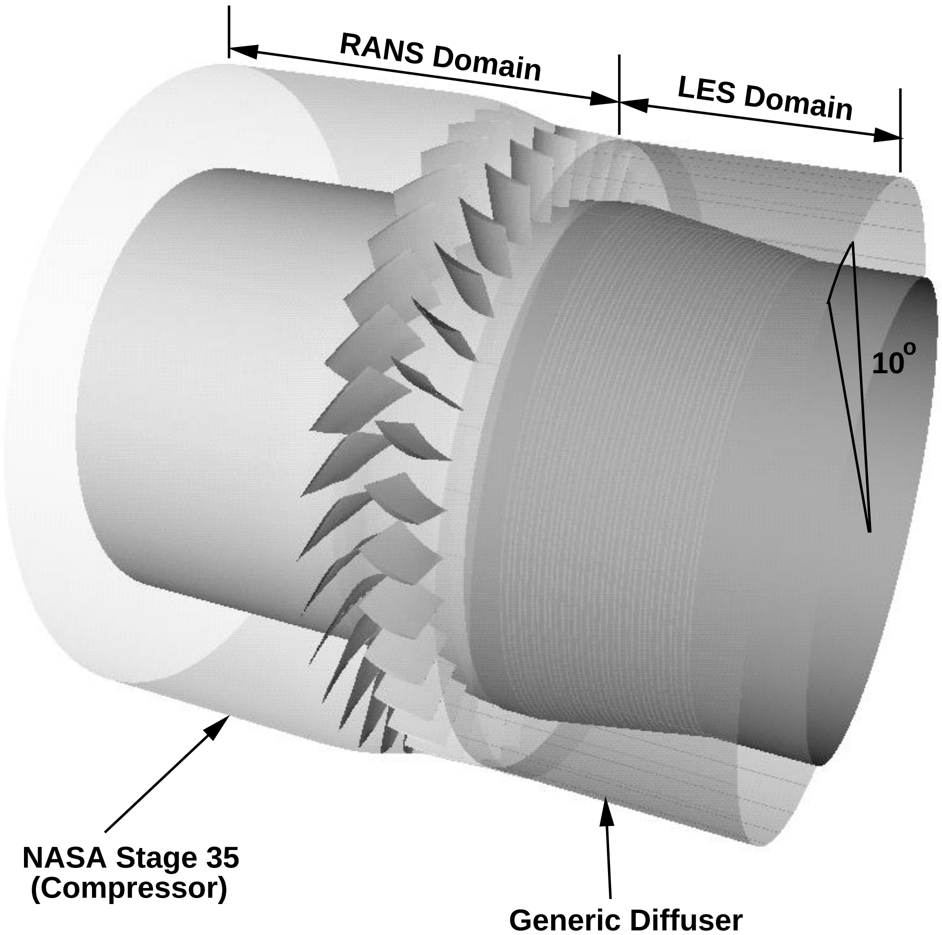

FIGURE 4. Geometry of the coupled NASA stage 35/prediffuser geometry. RANS domain includes one stage of a compressor, consisting of one rotor and one stator. LES domain includes the diffuser. A 10° axi-symmetric sector is computed.

computations in a turbomachinery environment and to identify practical issues involved in these calculations.

4.1. *Geometry*

The compressor geometry for the computed test-case corresponds to that of a modified NASA experimental rig stage 35. The 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 us 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 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, computations have been performed with two different LES flow solvers, a structured LES flow solver, which has been used already for many investigations of fundamental issues, and the CDP-α code. Since the structured flow solver is much faster than CDP, finer meshes can be used.

The structured LES flow solver is a code developed at the Center for Turbulence Research at Stanford by Pierce and Moin (1998). The filtered momentum equations with a low-Mach number assumption on an axi-symmetric structured single-block mesh are solved. A second-order finite-volume scheme on a staggered grid is used (Akselvoll & Moin 1996). The subgrid stresses are approximated with an eddy-viscosity approach, where the eddy viscosity is determined by a dynamic procedure (Germano *et al.* 1991; Moin *et al.* 1991).

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 structured LES flow solver consists of 3.5 million mesh points. The diverging wall is approximated with a stair-stepping function.

The LES mesh for the CDP flow solver consists of 500,000 control volumes and is concentrated near the walls. LES inflow boundary conditions for both flow solvers were defined corresponding to Eq. 3.1.

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.

4.2. *Results*

The computation using TFLO and the structured LES flow solver was carried out using 15 processors for the RANS domain and 3 processors for the LES domain. In total, 6 blade passings were computed. The computation was performed on an SGI Origin 3000 and needed 60 hours of wall-clock time.

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 $\approx 0.5\%$.

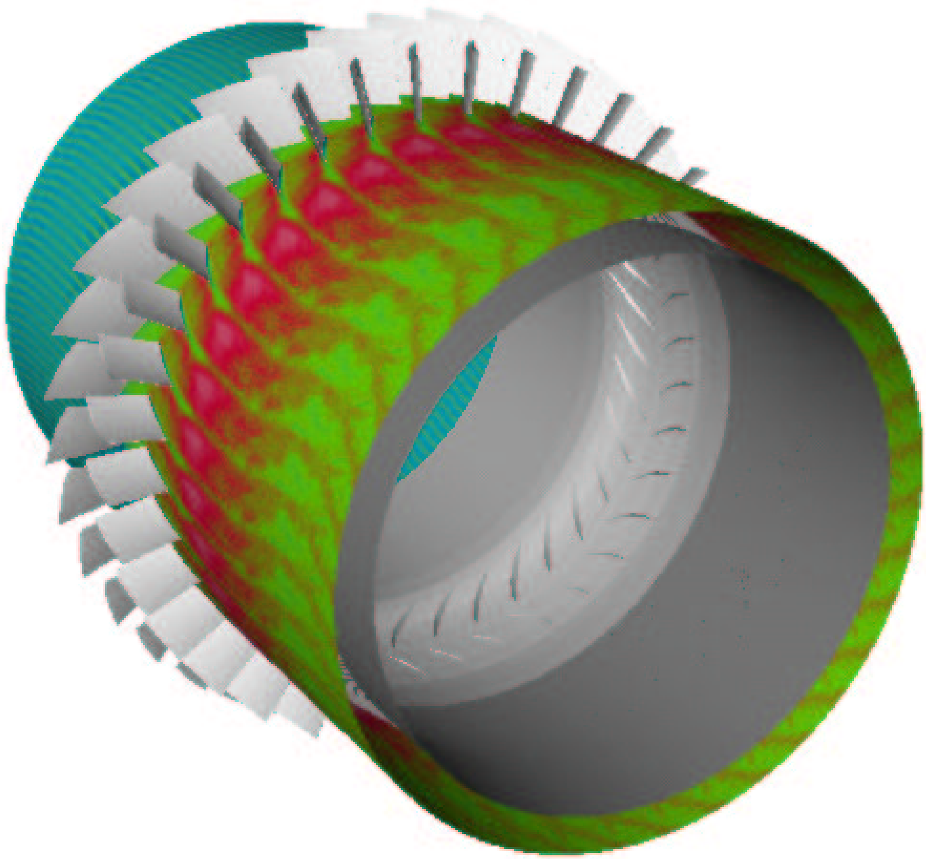

FIGURE 5. Integrated RANS-LES computation: velocity distribution in the NASA stage 35 - prediffuser geometry. The axisymmetric domain is copied and rotated around the circumference for visualization purposes.

The solutions of both LES flow solvers were nearly identical. Since the flow visualization using the structured LES flow solver is simpler, all pictures shown are results from the structured LES flow solver.

Figures 5 and 6 show the axial velocity distributions at 10% 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 10% span of the stator. Here the magnitude of the vorticity is depicted computed according to the unsteady flow field of both domains.

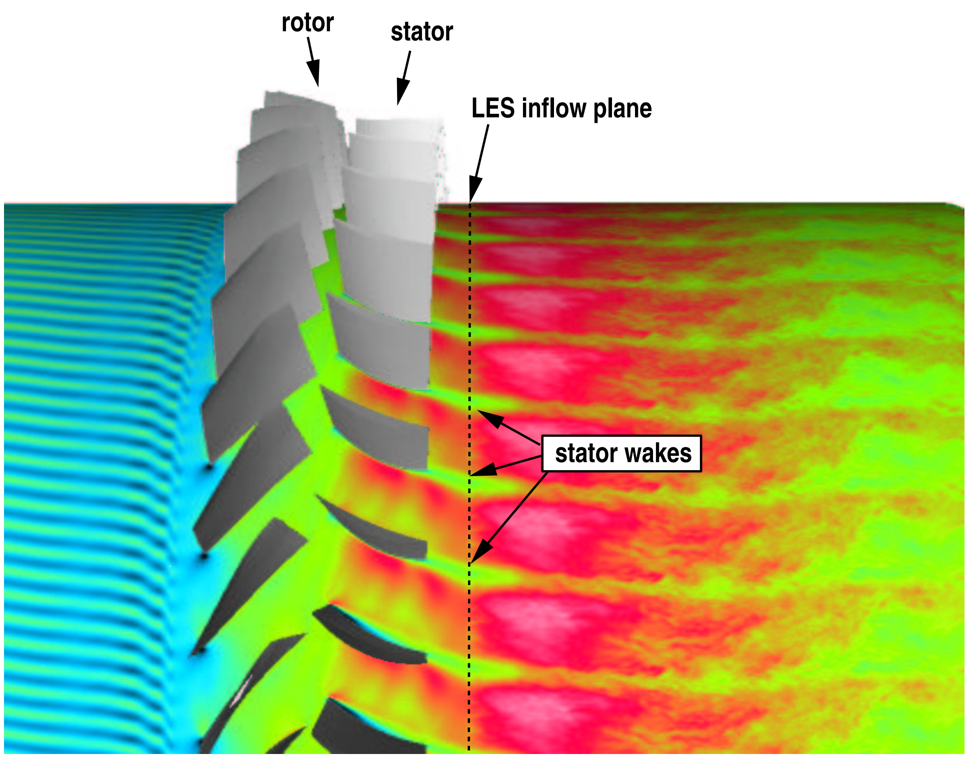

FIGURE 6. Integrated RANS-LES computation: velocity distribution in the NASA stage 35 - prediffuser geometry. Close-up of the interface.

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 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. 3.1.) 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 six 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.

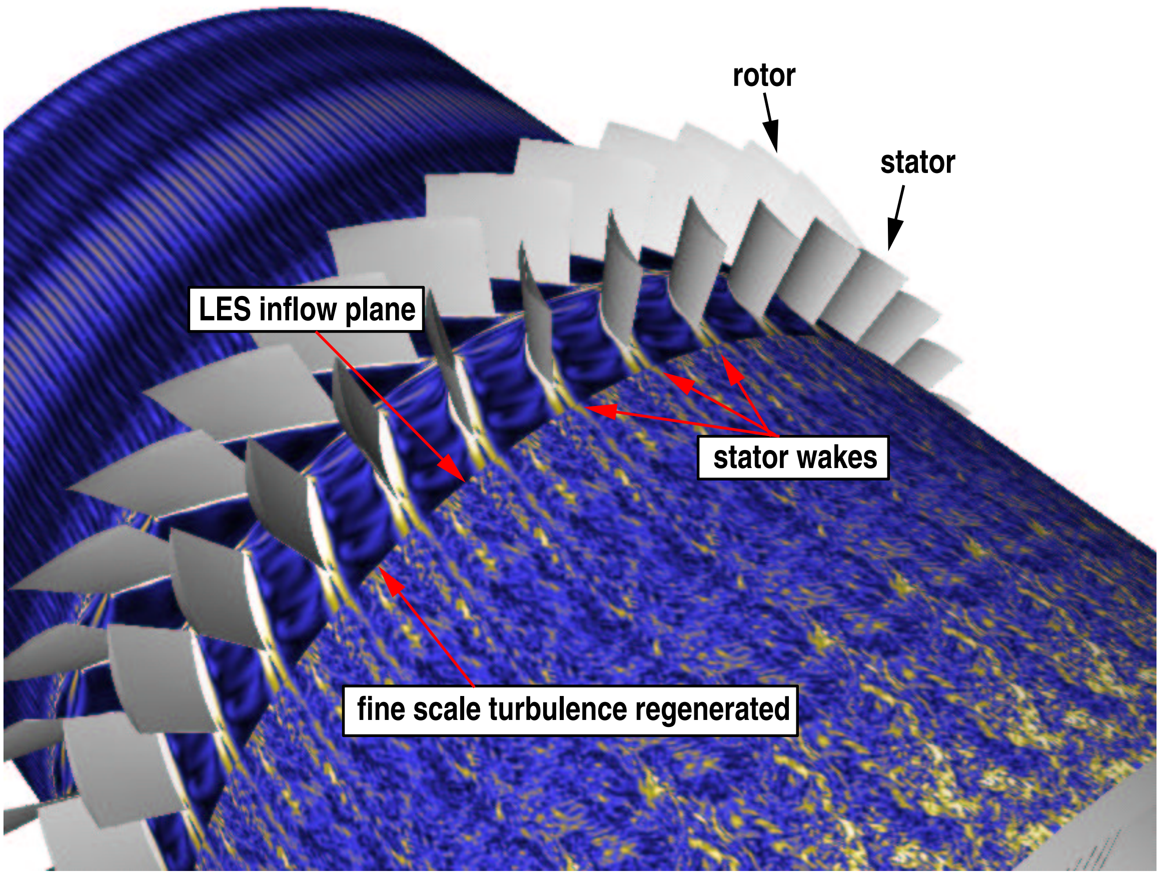

FIGURE 7. Integrated RANS-LES computation: Vorticity magnitude distribution in the coupled NASA stage 35/Prediffuser geometry. Vorticity created by the stator wake can be found in the LES domain.

5. Integrated Simulations: Conclusions

The computation of the coupled NASA stage 35/prediffuser geometry demonstrates the concept of integrated RANS-LES computations in a realistic environment. Future work will use the geometry of a real aircraft engine in order to characterize the flow in the prediffuser.

6. Acknowledgments

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

REFERENCES

AKSELVOLL, K., & MOIN, P. 1996 Large-eddy simulation of turbulent confined coannular jets. *J. of Fluid Mech.*, **315**, 387–411.

J. J. Alonso, L. Martinelli, and A. Jameson 1995 Multigrid unsteady Navier-Stokes calculations with aeroelastic applications. *AIAA Paper* 95-0048

Barker, A. G., and Carrotte, J. F. 2001a Influence of compressor exit conditions on combustor annular diffusers, Part 1: Diffuser performance. *Journal of Propulsion and Power*, **17**, No. 3, 678-686.

Barker, A. G., and Carrotte, J. F. 2001b Influence of compressor exit conditions on combustor annular diffusers, Part 2: Flow redistribution. *Journal of Propulsion and Power*, **17**, No. 3, 687-694.

A. Belov, L. Martinelli, and A. Jameson 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*

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, 2003.

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.

Davis, R., Yao, J.,Alonso, J. J., Paolillo, R., Sharma, O. P., 2003 Prediction of main/secondary-air system flow interaction in a high pressure turbine. *AIAA Paper* 2003-4833.

Dellenback, P. A., Metzger, D. E. & Neitzel, G. P 1988 Measurements in turbulent swirling flow through an abrupt axisymmetric expansion. *AIAA J.*, **26**, 669-681.

Germano, M., Piomelli, U., Moin, P. & Cabot, W., 1991 A dynamic subgrid-scale eddy viscosity model. *Phys. Fluids* A **(3)**, 1760-1765.

Jameson, A. 1991 Time dependent calculations using multigrid, with applications to unsteady flows past airfoils and wings. *AIAA paper* 91-1596

Mahesh, K., Constantinescu, G., Apte, S., Iaccarino, G. & Moin, P., 2001 Large-eddy simulation of gas turbine combustors *Annual Research Briefs 2001* Center for Turbulence Research, NASA Ames/Stanford Univ. 3-18.

Moin, P., Squires, K., Cabot, W. & Lee, S. 1991 A dynamic subgrid-scale model for compressible turbulence and scalar transport. *Phys. Fluids*, A **(3)**, 2746-2757.

Pierce, C. D. & Moin, P. 1998 Large eddy simulation of a confined coaxial jet with swirl and heat release. *AIAA Paper* 98-2892.

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

Schlüter, J. U., Pitsch, H., and Moin, P., 2002 Consistent boundary conditions for integrated LES/RANS simulations: LES outflow conditions. *AIAA paper* 2002-3121

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

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

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

Schlüter, J., Shankaran, S., Kim, S., Pitsch, H., Alonso, J. J., 2003d Integration of RANS and LES flow solvers for simultaneous flow computations. *AIAA paper*, (AIAA 2003-0085), 2003.

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

S. Shankaran, M.-F. Liou, N.-S. Liu, and R. Davis and J. J. Alonso 2001 A multi-code-coupling interface for combustor/turbomachinery simulations. *AIAA paper* 2001-0974

J. Yao, A. Jameson, J. J. Alonso, and F. Liu 2000 Development and validation of a massively parallel flow solver for turbomachinery flows. *AIAA paper* 00-0882