Center for Turbulence Research
Annual Research Briefs 2004

Integrated RANS-LES of a realistic gas turbine compressor/combustor assembly

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

1. Motivation and objectives

In the development of a gas turbine, 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 reasonable turnover time.

The goal of the Advanced Simulation and Computing (ASC) program 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 the entire flow path in an aircraft engine. However, considering the wide variety of the flow phenomena, which have to be simulated in the flow path of the engine, it is obvious that only the use of multiple specialized flow solvers, one for the turbo-machinery parts and one for the combustor, can guarantee appropriate efficiency and accuracy of a simulation. The reason for this is that the flow regimes and the turbulent scales vary dramatically in these two components. Most flow solvers used nowadays in the engine design process are specialized for one of the two tasks.

The flow field in the turbomachinery portions of the domain is characterized by both 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, including tip gaps and leakage flows. A number of flow solvers that have been developed to deal with this kind of problems have been in use in industry for many years. These flow solvers are typically based on the Reynolds-Averaged Navier-Stokes (RANS) approach. Due to the complexity of the flows in turbo-machinery, various parameters in the required turbulence models have to be adapted in order to provide accurate solutions. For turbomachinery flows, 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 greatly improved using flow solvers based on Large-Eddy Simulations (LES). While the use of LES increases the computational cost, LES has been the only predictive tool able to simulate these complex flows consistently. LES resolves the large scale turbulent motions in time and space and only the influence of the smallest scales, which are usually more universal and hence, easier to represent, has to be modeled (Ferziger 1996; Sagaut 2002). Since the energy containing part of the turbulent scales is resolved, a more accurate description of scalar mixing is achieved, leading to improved predictions of the combustion process (Veynante & Poinsot 1996). LES flow solvers have been shown in the past to be

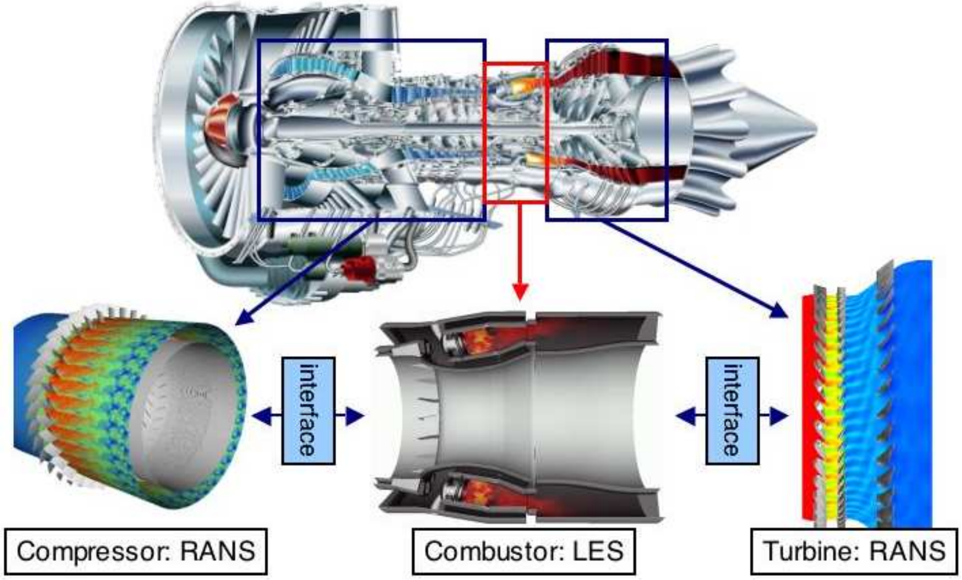

FIGURE 1. Decomposition of the engine for flow simulations. Compressor (Schlüter *et al.* 2004c) and turbine (Davis *et al.* 2002) with RANS; Combustor with (LESMoin & Apte 2004).

able to model simple flames and are currently adapted for use in gas turbine combustors (Poinsot *et al.* 2001; Constantinescu *et al.* 2003).

Here, we want to predict multi-component effects, such as compressor-combustor instabilities, combustor-turbine hot-streak migration and combustion instabilities. The flow solvers describing the different components in the gas turbine have to run simultaneously, each computing a part of the domain, and periodically exchanging flow information at the interface (Fig. 1). The simultaneous execution of multiple parallel flow solvers requires the definition of an interface which allows for the exchange of flow information and a framework for well-posed boundary conditions in order to process the exchanged data.

The approach to couple multiple simulation codes has already been applied in different fields, most notably in global climate simulations (Trenberth 1992), and found recently more attention in other areas of mechanical engineering (Adamis *et al.* 1998). However, coupling RANS and LES flow solvers is a very recent approach and a unique method to construct an LES-RANS hybrid. While other LES-RANS hybrid approaches, such as Detached-Eddy Simulations (DES) (Spalart 2000) and Limited-Numerical Scales (LNS) (Batten *et al.* 2002) combine LES and RANS in a single flow solver, the approach to couple two existing flow solvers has the distinct advantage to build upon the experience and validation that has been put into the individual codes during their development, and also to run simulations in different domains at different time-steps.

In the current study we are presenting the coupling approach and apply it to a compressor-prediffuser geometry of a real Pratt & Whitney aircraft gas turbine engine. The interface between compressor and combustor constitutes the upstream interface of a full engine simulation (Fig. 1). The flow leaving the compressor enters first into the prediffuser of the combustor. The function of the prediffuser is to decelerate the flow with a maximum of pressure gain (Klein 1995). For this reason, prediffusers are operated close to the point of flow separation. The flow conditions in the prediffuser ultimately influence the flow split in the combustor and determine the amount of air entering the combustion chamber through the fuel injector. Although the performance of the diffuser is influenced by the flow field leaving the compressor (Barker & Carrotte 2001a; Barker & Carrotte 2001b), little is known about the exact flow features at this location during the design phase of an engine. The reason for this is that the two components are usually developed in isolation and combined tests are done only in the final prototype assembly.

Here, we will apply the approach of multiple flow solvers to study the flow interactions between these two components. A RANS flow solver computing the final stage of the compressor is coupled with an LES flow solver computing the combustor. The flow in the turbomachinery parts is compressible and governed by the flow around the blades. Hence, a RANS flow solver is an appropriate tool to assess the flow in this section. On the other hand, the prediction of flow separation is facilitated in the LES approach. And while the flow in the current design is not separated, predictions of design modifications have to be able to assess these flow features accurately.

The present paper is organized in the following way:

(*a*) Firs, we describe the RANS and LES flow solvers as well as the interface and the boundary conditions.

(*b*) Then, the application of this approach to a generic compressor/diffuser geometry is shown.

(*c*) Finally, the approach is applied to a real Pratt & Whitney gas turbine engine geometry.

2. Flow Solvers and Interface

In the following we briefly present the computational framework of this study consisting of the flow solvers and the interface. A more comprehensive description of the interface can be found in Schlüter *et al.* (2003b) and Schlüter *et al.* (2004d).

2.1. *RANS Flow Solver*

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, 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 (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.

2.2. *LES Flow Solver*

The LES flow solver used for the current study is the CDP 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.

2.3. *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 challenges encountered. In previous work interface routines have been established and validated with simple geometries (Shankaran *et al.* 2001; Schlüter *et al.* 2003c; Schlüter *et al.* 2004d).

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.* (2003a) and Schlüter *et al.* (2004d).

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.

2.4. *Boundary Conditions*

The definition of the boundary conditions requires special attention, especially on the LES side, due to the different physical modeling approaches. Since on the LES side a part of the turbulent energy 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 (Schlüter *et al.* 2002a; Schlüter *et al.* 2004b).

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 by scaling the data base solution to the RANS mean profiles and velocity fluctuations (Schlüter *et al.* 2004a).

On the RANS side, inlet and exit boundary condition are applied using the time-averaged solution from the LES side. More advanced boundary conditions are under investigation (Kim *et al.* 2004).

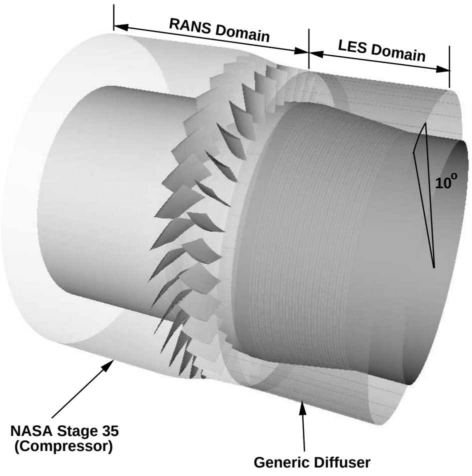

FIGURE 2. Geometry of coupled NASA stage 35/prediffuser domain.

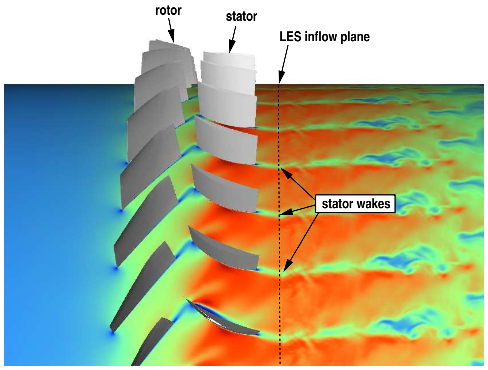

FIGURE 3. Integrated RANS-LES of compressor/prediffuser: Velocity distribution at the 50% plane. Close-up of the interface.

2.5. *Validation*

In the past, we have reported the validation of the flow solvers and the interface. The multi-block RANS flow solver TFLO has been in use for turbomachinery applications for several years. The description of the flow solver and some validation studies have been reported by Yao *et al.* (Yao *et al.* 1998; Yao *et al.* 2000). The unstructured LES flow solver CDP has been validated separately with a focus on combustor simulations (Ham *et al.* 2003; Moin & Apte 2004; Wu *et al.* 2004). The validation for the interface has been performed for integrated simulations on simplified geometries (Schlüter & Pitsch 2002b; Schlüter *et al.* 2003b; Schlüter *et al.* 2004d). The necessary boundary conditions for integrated simulations have been validated in detail (Schlüter *et al.* 2004a; Schlüter *et al.* 2004b; Kim *et al.* 2004).

3. Integrated RANS-LES of the NASA Stage 35/Diffuser

In this section we demonstrate the value of coupled RANS-LES computations for gas turbine applications. The test-case is that of the NASA Stage 35 compressor that we extended behind the stators with a diffuser. This geometry has been studied by Schlüter *et al.* (2003d ; 2004c). Previous studies have demonstrated the capability of the integrated RANS-LES approach to simulate this kind of geometry. Here, we want to focus on demonstrating the advantages such an approach may give in comparison to simulations of the single components.

3.1. *Geometry*

The compressor geometry for the computed test-case corresponds to that of a modified NASA Stage 35 experimental rig, which consists of a rotor with 46 rotor blades and a stator with 36 stators vanes. 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.

On the LES side, we use the structured LES flow solver developed at CTR. A more comprehensive description of the geometry, the mesh topology and the flow conditions, as well as some results on the main flow features (see also Fig. 3) can be found in Schlüter *et al.* (2003d).

Here, we assess the value of integrated RANS-LES simulations for this geometry. We will perform an integrated RANS-LES simulation of the entire domain. Then, we will use the computed flow field at the inlet of the diffuser to define the inflow boundary conditions for a separate, uncoupled LES simulation of the diffuser. The comparison of the integrated RANS-LES with the separate LES will give an insight on the importance of the flow development in the compressor on the diffuser flow.

3.2. *Results*

The integrated RANS-LES computations were carried out using 64 processors for TFLO and 16 processors for the structured LES flow solver. Eight blade passings were computed in 60 hours of wall clock time using an IBM Power3. The uncoupled LES computations were performed on 32 processors in 10 hours wall clock time computing a physical time-span equivalent to 5 blade passings.

Figure 4 compares the flow development in the diffuser for the two different simulations. The solid lines represent the uncoupled LES computation. The inlet velocity profile and the level of turbulence has been specified according to the time-averaged RANS solution at the outlet of the compressor. This solution has been retrieved from the integrated solution and is used to specify the inlet boundary conditions of an the uncoupled LES computation. The turbulence in this inlet plane is added to the mean velocity profile using the method of Schlüter *et al.* (2004d) using the identical turbulence inflow data base as in the integrated RANS-LES. The profiles of velocity fluctuations contain the turbulence modeled by the RANS turbulence model as well as the long-wave flow modulation, which is resolved in the RANS computation. The dashed lines are from the LES domain of the coupled RANS-LES computation, which means, that at each RANS time step the LES inflow is updated according to the unsteady solution in the compressor.

Comparing the velocity profiles at the inlet plane, we can see that both solutions are identical. However, further downstream both solutions are distinctively different.

The profiles of the velocity fluctuations show a similar behavior. At the inlet, both profiles are identical. Here, already shortly downstream, the velocity fluctuations are much larger in the integrated RANS-LES computation. This can be explained with the fact that in the integrated RANS-LES unsteady flow features from the compressor are transfered to the LES. This results in temporarily stronger gradients. The production of turbulence is determined as: $P = \overline{u_i' u_j'} \frac{\partial \bar{u}_i}{\partial x_j}$ and depends on these gradients. Hence, in the coupled RANS-LES simulation the RANS simulation does not only provide turbulent energy in its turbulence model, but also the potential to create more turbulence in the unsteady mean velocity gradients. In the current case, the additional turbulence production leads to a different turbulence field which results in a different mean flow field than in the uncoupled LES computation. This demonstrated that in the current case of the NASA Stage 35/diffuser the use of an integrated RANS-LES can improve the prediction of the diffuser flow.

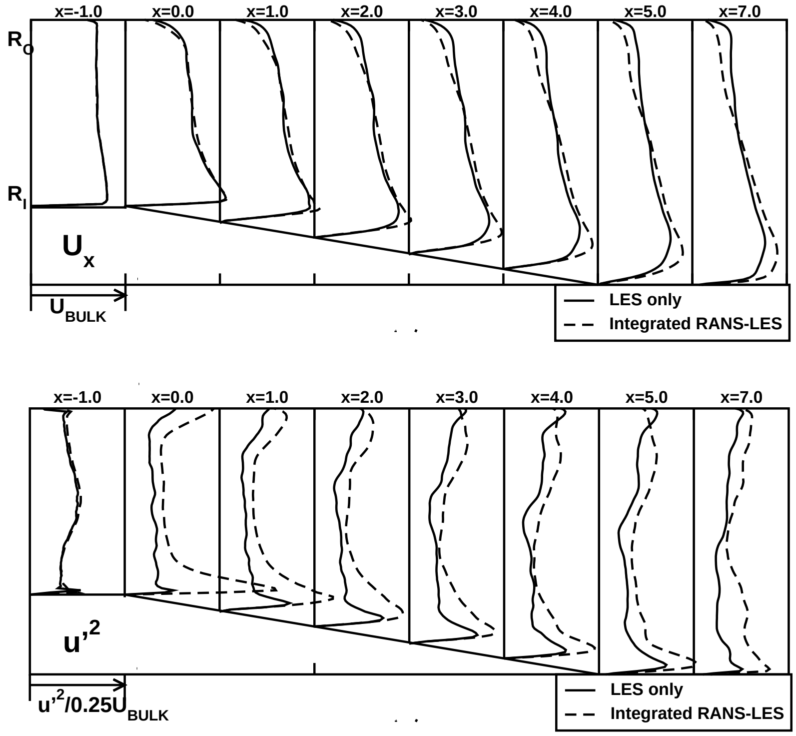

FIGURE 4. Velocity profiles in the diffuser; Solid lines: LES only; Dashed lines: RANS-LES; Above: axial velocity; Below: axial velocity fluctuations

3.3. *Pratt & Whitney Engine Geometry*

In the previous sections we presented the coupled RANS-LES approach and its application to a simplified compressor geometry. Here, we want to demonstrate this approach using a real engine geometry.

The geometry considered is that of a Pratt & Whitney aircraft engine (Fig. 5 & 6). Here, we present a simulation of the last stage of the high pressure compressor consisting of one rotor and the exit guide vanes (EGV) using the RANS approach. This RANS simulation is coupled with a LES of the prediffuser and the entire combustor. We chose to simulate the entire combustor including the fuel injector, since the flow blockage by the fuel injector and the resulting flow split is considered to be important for the performance of the diffuser (Barker & Carrotte 2001a; Barker & Carrotte 2001b; Klein 1995). However, the flow in the combustion chamber is non-reactive. The computation of reactive flows has been already demonstrated for this geometry (Moin & Apte 2004), but we consider it as not necessary for the purpose of the present demonstration.

The geometry is a 20° segment of the full engine geometry, which means that we compute one fuel injector. The blade count of the last stage of the compressor was rescaled to fit the 20° segment, and four rotor blades and seven exit guide vanes are computed in total. The RANS mesh consists of 500,000 cells in a structured multi-block mesh.

The complexity of the combustor geometry requires the use of the unstructured LES flow solver CDP. The combustor mesh consists of 3,000,000 unstructured mesh cells and

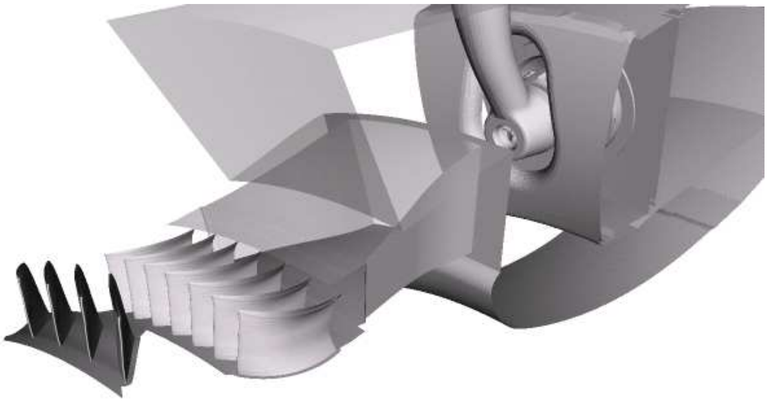

FIGURE 5. Geometry of Pratt & Whitney test case: In the front, the segment of the compressor (4 rotor blades, 7 exit guide vanes) is shown. The flow leaving the compressor enters the prediffuser box before it enters a plenum in front of the fuel injector.

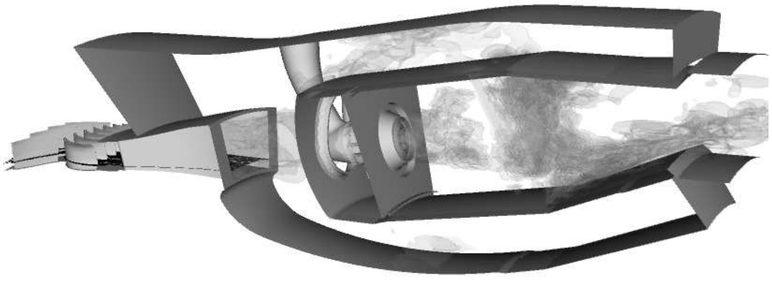

FIGURE 6. Geometry and flow visualization in the combustor. Note the compressor stage upstream of the diffuser. Smoke visualization demonstrates flow features of the cold flow.

is refined in the diffuser part. The mesh resolution in the prediffuser was chosen to correspond approximately to the coarse mesh resolution of the diffuser simulation (Wu *et al.* 2004).

The computation of 10 blade passings was performed using 128 processors on an IBM SP3. One blade passing needed 10 hours wall clock time. The entire computation was performed within one week.

Figures 7 and 8 show a flow visualization of this computation. For this visualization, the computed 20° segment is shown several times in different azimuthal locations in order to present a picture of a 360° engine. In the combustor, isosurfaces of the axial velocity demonstrate the level of detail of the flow simulation in the combustor. In the compressor and the prediffuser, a clip plane at 50% span of the exit guide vane (EGV) shows the axial velocity distribution. The most dominant flow feature at the interface is the propagation of the wakes of the EGV into the prediffuser. The wakes create large scale turbulent structures inside the prediffuser. Figure 9 shows a close-up of the axial velocity contours at the interface.

This computation of the Pratt & Whitney real engine geometry is the first of its kind. Simulations of this portion of a gas turbine usually encounter difficulties, either in the description of the compressor flow, which can be computed only by a RANS approach, or the representation of the diffuser flow, which needs to capture the possible presence of flow detachment accurately and is largely improved by the use of LES.

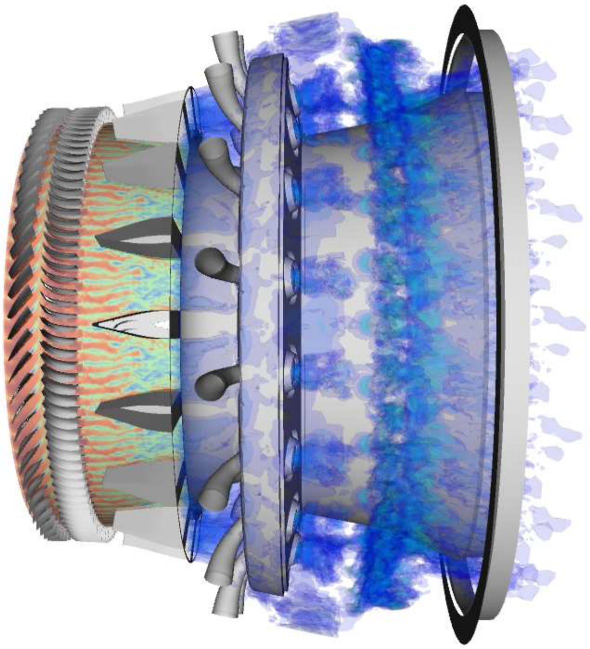

FIGURE 7. Isocontours of the axial velocity at the 50% plane in the compressor and diffuser. Isosurfaces of axial velocity in the combustor. A 20° segment is computed.

The current state-of-the-art in diffuser simulations uses either radial profiles obtained from experiments or steady state data from uncoupled RANS simulations of the compressor. The integrated simulation allows not only to obtain a more precise representation of the compressor flow, but also to simulate the geometries accurately without the need of input from experiments.

4. Conclusions

In this study we presented an approach to couple two separate flow solvers, one based on the RANS approach, the other based on LES, to improve flow predictions of complex flows. As an example, we investigated the flow leaving the compressor and entering the diffuser. We have validated the interface and the flow solvers extensively in previous studies. Here, we applied the multi-code coupling approach to a compressor-combustor geometry.

First, a computation of a simplified compressor/diffuser geometry demonstrated the basic flow features of such a geometry and showed the value of coupled RANS-LES for this application.

Then, the approach was applied to a real engine geometry. Basic flow features of this flow configuration were identified.

The integrated RANS-LES environment provides a computational test bench for the assessment of complex flow interactions, such as those of the compressor/combustor coupling in an aircraft gas turbine engine.

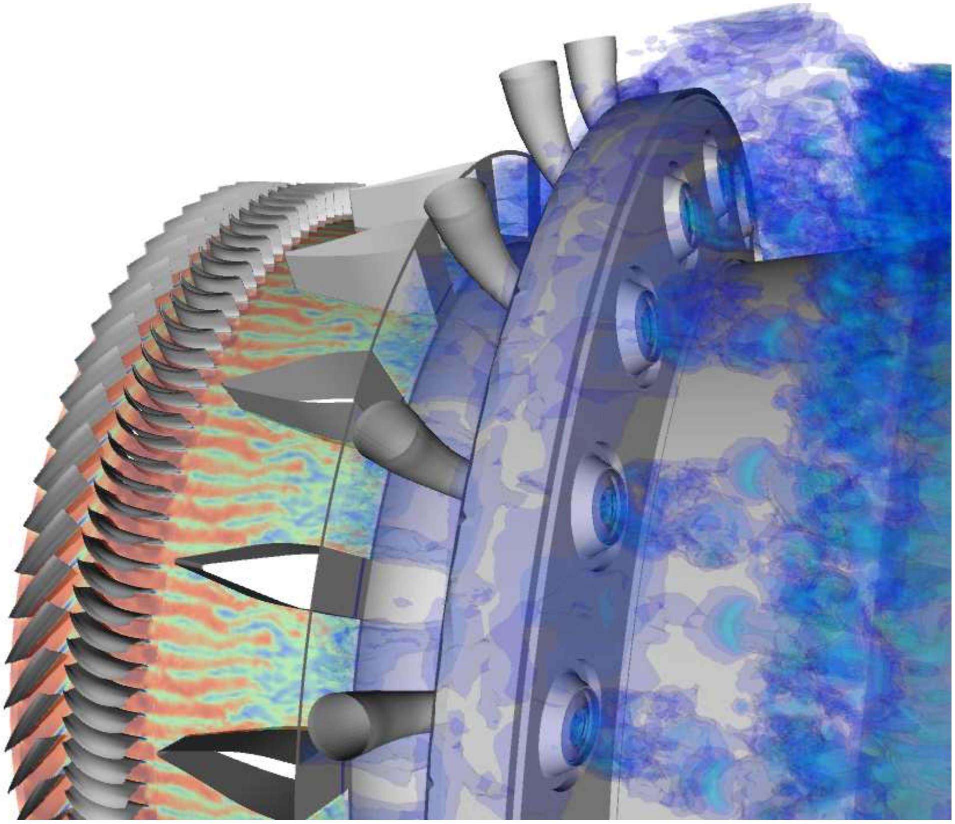

FIGURE 8. Close-up: Isocontours of the axial velocity at the 50% plane in the compressor and diffuser. Isosurfaces of axial velocity in the combustor. A 20° segment is computed.

5. Acknowledgments

We wish to thank the US Department of Energy for the support under the ASC program.

We also thank Dr. Saadat Syed, Dr. Jinzhang Feng and Dr. Stephen Krautheim from Pratt & Whitney for providing the engine geometry, helpful comments and discussions.

REFERENCES

ADAMIDIS P., BECK, A., BECKER-LEMGAU, U., DING, Y, FRANZKE, M., HOLTHOFF, H., LAUX, M., MÜLLER, A. MÜNCH, M., REUTER, A., STECKEL, B., TILCH, R. 1998 Steel strip production - a pilot application for coupled simulation with several calculation systems. *Journal of Materials Processing Technology*, (80-81):330–336, 1998.

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

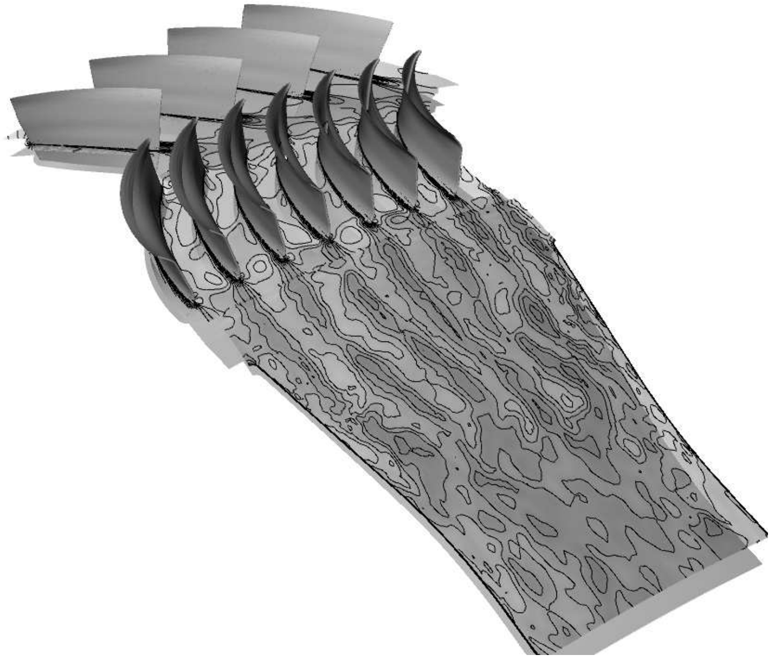

FIGURE 9. Isocontours of the axial velocity at the 50% plane in the compressor and diffuser. Close up of the interface.

on combustor annular diffusers, Part 2: Flow redistribution. *Journal of Propulsion and Power*, **17**, No. 3, 687-694.

BATTEN, P., GOLDBERG, U., CHAKRAVARTHY, S. 2002 LNS-an approach towards embedded LES. *AIAA paper*, (AIAA 2002-0427), 2002. 40th AIAA Aerospace Sciences Meeting and Exhibit, January 2002, Reno, NV.

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.

FERZIGER, J. H. 1996 *New Tools in Turbulence Modelling*, volume New Tools in Turbulence Modelling of *Les edition physique*, chapter 2 Large eddy simulation: an introduction and perspective, pages 29–47. Springer, 1996.

GERMANO, M., PIOMELLI, U., MOIN, P. & CABOT, W., 1991 A dynamic subgrid-scale eddy viscosity model. *Phys. Fluids* A **(3)**, 1760-1765.

HAM, F., APTE, S., IACCARINO, G., WU, X., HERRMANN, M., CONSTANTINESCU, G., MAHESH, K., MOIN, P. 2003 Unstructured LES of reacting multiphase flows in realistic gas turbine combustors. *CTR Annual Research Briefs*, pages 139–160, 2004. Center for Turbulence Research, Stanford.

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

KIM, S., SCHLÜTER, J. U., WU, X., ALONSO, J. J., PITSCH, H. 2004 Integrated simulations for multi-component analysis of gas turbines: RANS boundary conditions. *AIAA Paper*, (AIAA 2004-3415), July 2004.

KLEIN, A. 1995 Characteristics of combustor diffusers. *Prog. Aerospace Sci.*, 31:171–271, 1995.

MOIN, P., APTE, S. 2004 Large-eddy simulation of realistic gas turbine combustors. *AIAA Paper*, (AIAA 2004-0330), January 2004.

POINSOT, T., SCHLÜTER, J., LARTIGUE, G., SELLE, L., KREBS, W., HOFFMANN, S. 2001 Using large eddy simulations to understand combustion instabilities in gas turbines. *IUTAM Symposium on Turbulent Mixing and Combustion*, pages 1–8, 2001. Kingston, Canada 2001.

SAGAUT, S. 2002 *Large eddy simulation for incompressible flows.* Springer, Berlin, 2 edition, 2002.

SCHLÜTER, J. U., PITSCH, H., MOIN, P., 2002A Consistent boundary conditions for integrated LES/RANS simulations: LES outflow conditions. *AIAA paper* 2002-3121

SCHLÜTER, J., PITSCH, H. 2002B Integration of LES and RANS flow solvers: Interface validation. *CTR Annual Research Briefs*, 2002. Center for Turbulence Research, Stanford.

SCHLÜTER, J. U., PITSCH, H., MOIN, P., 2003A Boundary conditions for LES in coupled simulations. *AIAA paper*, (AIAA-2003-0069), 2003.

SCHLÜTER, J., SHANKARAN, S., KIM, S., PITSCH, H., ALONSO, J. J., 2003B 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.

SCHLÜTER, J., SHANKARAN, S., KIM, S., PITSCH, H., ALONSO, J. J., 2003C Integration of RANS and LES flow solvers for simultaneous flow computations. *AIAA paper*, (AIAA 2003-0085), 2003.

SCHLÜTER, J. U., WU, X., KIM, S., ALONSO, J. J., PITSCH, H. 2003D Integrated RANS-LES computations of turbomachinery components: Generic compressor/diffuser. *CTR Annual Research Briefs*, pages 357–368, 2003.

SCHLÜTER, J. U., PITSCH, H., MOIN, P., 2004A Large eddy simulation inflow conditions for coupling with Reynolds-averaged flow solvers. *AIAA Journal*, 42(3):478–484, March 2004.

SCHLÜTER, J. U., PITSCH, H., MOIN, P., 2004B Outflow conditions for integrated Large-Eddy simulation/Reynolds-averaged Navier-Stokes simulations. *accepted for publication to AIAA Journal*, 2004.

SCHLÜTER, J. U., WU, X., KIM, S., ALONSO, J. J., PITSCH, H. 2004C Integrated RANS-LES computations of gas turbines: Compressor-diffuser. *AIAA Paper*, (AIAA 2004-0369), January 2004.

SCHLÜTER, J. U., WU, X., KIM, S., ALONSO, J. J., PITSCH, H. 2004D A framework for coupling Reynolds-averaged with large eddy simulations for gas turbine applications. *Journal of Fluids Engineering*, submitted, 2004.

SHANKARAN, S., LIOU, M.-F., LIU, N.-S., DAVIS, R., ALONSO, J. J. 2001 A multi-code-coupling interface for combustor/turbomachinery simulations. *AIAA paper*, (AIAA 2001-0974), 2001.

SMAGORINSKY 1963 General circulation experiments with the primitive equations, I, the basic experiment. *Mon. Weather Rev.*, 91(3):99–152, 1963.

SPALART 1963 Trends in turbulence treatments. *AIAA paper*, (AIAA 2000-2306), 2000. Fluids 2000, June, Denver, CO.

TRENBERTH, K. E. 1992 *Climate System Modeling.* Cambridge, New York, 1992. chapter 9.

VEYNANTE, D., POINSOT, T. 1996 Reynolds averaged and large eddy simulation modeling for turbulent combustion. In *New Tools in Turbulence Modelling*, Les edition physique. Springer, pp 105–140

WU, X., SCHLÜTER, J., MOIN, P., PITSCH, H., IACCARINO, G, HAM, F., 2004 Identification of an internal layer in a diffuser. *CTR Annual Research Briefs*, 2004. Center for Turbulence Research, Stanford.

YAO, J., JAMESON, A., ALONSO, J. J., LIU, F. 2000 Development and validation of a massively parallel flow solver for turbomachinery flows. *AIAA paper* 00-0882

YAO, J., JI, S., LIU, F., JENNIONS, I. K. 1998 Computation of a steady flow in a turbomachinery stage by third-order accurate high-resolution scheme. *AIAA paper* 98-3561