Large-scale integrated LES-RANS simulations of a gas turbine engine

By J. Schlüter, S. Apte, G. Kalitzin, E. v. d. Weide, J. J. Alonso AND H. Pitsch

1. Motivation and objectives

Today's use of Computational Fluid Dynamics (CFD) in gas turbine design is usually limited to component simulations. The demand on the models to represent the large variety of physical phenomena encountered in the flow path of a gas turbine mandates the use of a specialized and optimized approach for each component.

The flow field in the turbomachinery portions of the domain is characterized both by high Reynolds-numbers and by 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. Here, the unsteady flow field is ensemble-averaged, thereby removing all dependence on the details of the small scale turbulence. A turbulence model becomes necessary to represent the portion of the physical stresses that has been removed during the averaging process. Due to the complexity of the flows in turbomachinery, various parameters in these turbulence models have to be adapted in order to deliver accurate solutions. Since this kind of flow has been the subject of a large number of investigations, 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 consistently these complex flows. 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 (Raman & Pitsch, 2005). LES flow solvers have been shown in the past to be able to model simple flames and are currently adapted for use in gas turbine combustors (Poinsot *et al.*. 2001, Constantinescu *et al*, 2003).

An attempt to simulate multi-component effects so far required integrating the two approaches into a single solver. Here, we want to demonstrate an alternative, more flexible strategy. We have developed a software and an environment that allows for the simultaneous execution of multiple solvers. Each of these solvers computes a portion of a given flow domain and exchanges flow data at the interfaces with its peer solvers. We will demonstrate this approach in a simulation of a 20 degree sector of the high pressure part of a gas turbine engine. We will show that such a simulation can deliver accurate results
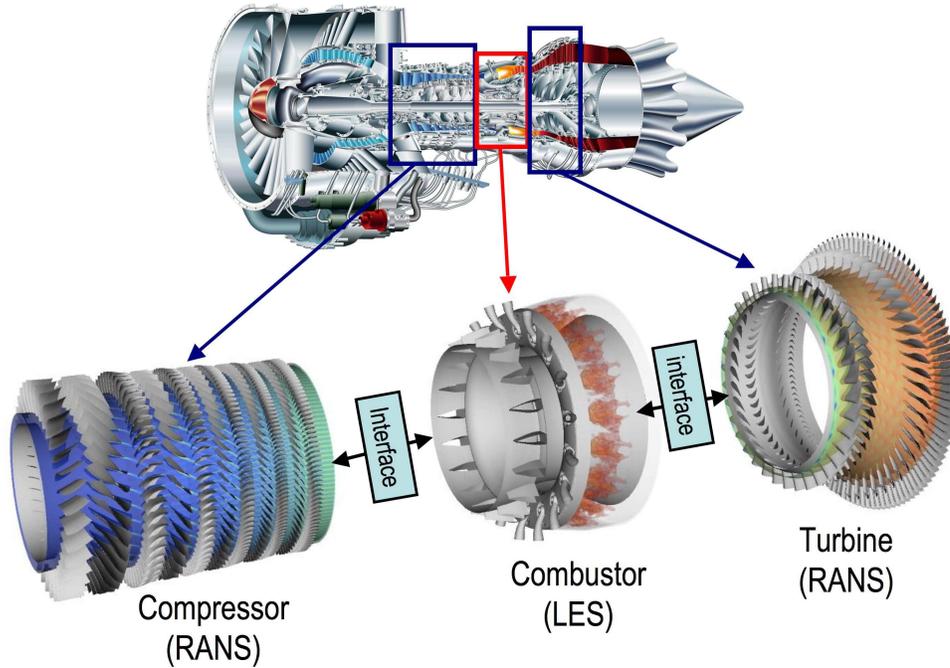
FIGURE 1. Decomposition of the engine for flow simulations. Compressor and turbine with RANS; Combustor with LES.

within a reasonable turnover time, which is necessary for use in the design process of an engine.

The approach of coupling multiple simulation codes has already been applied in other areas of application, most notably in global climate simulations (Trenberth, 1992), and has recently drawn more attention in other areas of mechanical engineering (Adamidis *et al.* 1998). However, the idea to couple RANS and LES flow solvers is a very recent approach and a unique method for constructing 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 of coupling two existing flow solvers has the distinct advantage of building upon the experience and validation that has been put into the individual codes during their development. It provides the possibility running simulations in different domains at different time-steps and provides a higher degree of flexibility.

In the following, we will present the approach, some details of the applied solvers, and the coupling software. Furthermore, we will demonstrate the concept in a simulation of the flow in the high-pressure part of a Pratt & Whitney gas turbine engine.

2. Solver framework

2.1. *RANS flow solver*

The RANS flow solver used for this investigation is the SUmb 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 for 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 (Germano *et al*, 1991).

2.3. *Boundary conditions*

The definition of the boundary conditions requires special attention, especially on the LES side due to the different turbulence modeling approaches. Since on the LES side a part of the turbulent spectrum is resolved, the challenge is to regenerate and preserve the turbulence at the boundaries. To impose RANS solutions at the outflow of the LES domain, a body force method has been developed (Schlüter *et al*, 2005a).

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 lacks the temporal and spatial correlations of real turbulence. Such random fluctuations are therefore dissipated very quickly. Instead, a database 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 (Schlüter *et al*, 2004)

$$u_{i,\text{LES}}(t) = \underbrace{\overline{u}_{i,\text{RANS}}(t)}_{I} + \underbrace{(u_{i,\text{DB}}(t) - \overline{u}_{i,\text{DB}})}_{II} \cdot \underbrace{\frac{\sqrt{\overline{u'^2_{(i)}}}_{\text{RANS}}(t)}{\sqrt{\overline{u'^2_{(i)}}}_{\text{DB}}}}_{III} , \tag{2.1}$$

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 (2.1) 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, inlet and exit boundary conditions are applied using the time-averaged solution from the LES side (Kim *et al.*, 2004).
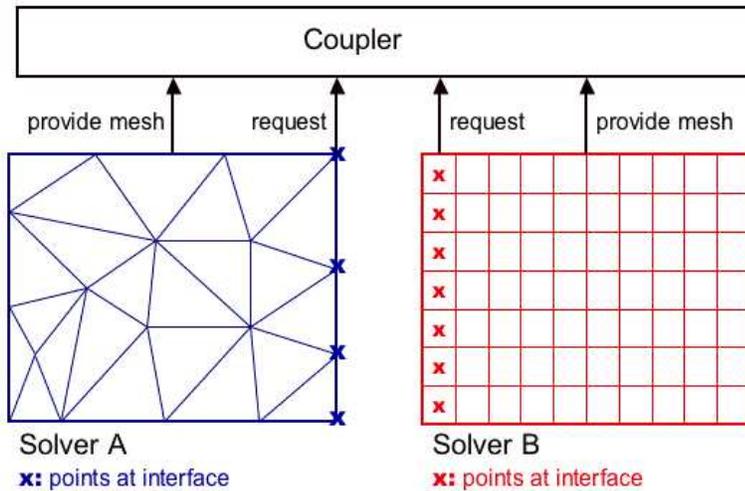
FIGURE 2. CHIMPS approach: solvers communicate location of their interface points and their mesh and solution to the coupler. The coupler determines how to provide information to the solver at the interface nodes.

2.4. *Interface*

Previous approaches to couple solvers were based on a pure MPI approach (Shankaran *et al*, 2001, Schlüter *et al*, 2003a, Schlüter *et al*, 2003b, Schlüter *et al*, 2005b). In this approach, MPI is used to allow the solvers to communicate directly with each other. This requires that in each of the solvers, algorithms have to be implemented that perform the tasks associated with the coupling.

However, the complexity of real engineering applications requires a more robust and a more user-friendly way to couple individual solvers. Here, instead of implementing all coupling routines (communication, search, and interpolation) in all solvers separately, we have developed a separate software module that performs these tasks and facilitates the coupling process. The idea is to remove some of the workload from the solvers, especially the search, interpolation, and communication routines. The solvers communicate with the coupler software only (Fig. 2) and not with each other. In this setup, the individual solvers require no information about other participating solvers. Instead, each solver only requests and receives data for its own interfaces from the coupler. The coupler performs all searches and interpolations. In order to be able to perform these tasks, the coupler requires the meshes and the solutions of the solvers.

The coupling software module that we have developed is called **C**oupler for **H**igh-Performance **I**ntegrated **M**ulti-**P**hysics **S**imulations (CHIMPS) (Schlüter *et al*, 2005c). It is based on the script language python.

The script language python, together with its parallel version pyMPI, allows for the simplification of the communication between the solvers and CHIMPS. Instead of defining MPI calls, python functions are defined, which allows more freedom in the implementation. The communication is then handled like a function call with arguments, with the data being passed in the argument list.

The advantage of the module is that it is written in a general fashion, and solvers can be adapted easily to communicate with other solvers. The software module performs many of the required coupling tasks such as searches, interpolations, and process-to-
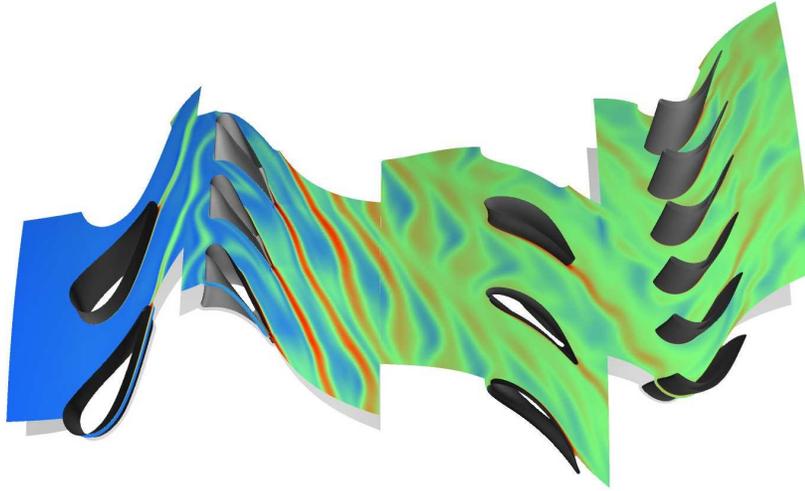
FIGURE 3. Turbine, 20 degrees sector; Instantaneous entropy distribution for the unsteady solution in a radial grid plane for the scaled geometry.

process communication. The adaptation of solvers to the coupling module is facilitated, compared with previous approaches.

3. Component simulations

In this investigation, we will demonstrate the application of this method in a multi-component integrated simulation of a Pratt & Whitney aircraft engine. To assess component interactions, the results should ultimately be compared with simulation results for the individual components. For this purpose, we will first briefly present simulations of the high-pressure compressor, the combustor, and the high-pressure turbine. The results of the component simulations will also be used as initial conditions in the multi-component simulations.

3.1. *Turbomachinery simulations*

The blade counts in turbomachinery are normally such that no sector periodicity occurs. This is done to avoid instabilities caused by resonance between two components. As a consequence, the true unsteady simulation can only be done for the entire wheel, unless simplifying assumptions are made. The currently accepted practice is to rescale the blade counts of the turbomachinery stages such that sector periodicity is obtained. To preserve the same flow blockage, the pitch of the blades is adjusted according to common industry practice.

Here we have chosen to simulate a 20 degree sector of the engine. In view of the full engine simulation, this is the smallest sector that can be chosen, since it contains one fuel injector. The rescaling and the pitch adjustment were performed following current industry practice.

The unsteady simulations for both the compressor and the turbine are started from the mixing plane solutions. The second order implicit time integration scheme is used. The resulting nonlinear system of equations is solved using the dual time stepping procedure (Jameson, 1991), in which 25 3W multigrid cycles are used per physical time step. For a full wheel revolution, 6,300 time steps are used for the compressor and 2700 time
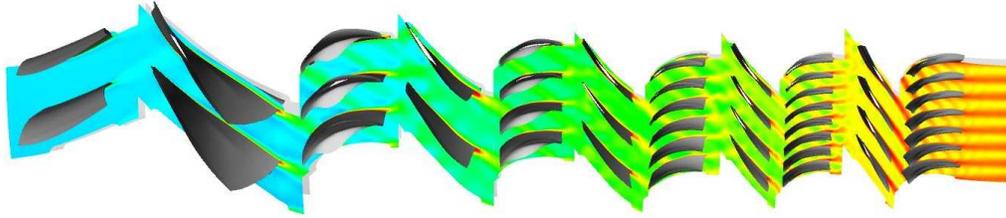
FIGURE 4. Compressor, 20 degrees sector; instantaneous entropy distribution for the unsteady solution in a radial grid plane for the scaled geometry.

steps are used for the turbine. This correspond to 50 time steps for a blade passing of the blade row with the highest blade count and the highest rotational speed. Figure 4 presents the instantaneous entropy distribution after 1500 time steps, which corresponds to approximately 85 degrees of rotation. Detailed analysis of the results is currently under way.

3.2. *20 degree sector combustor simulations*

In addition, a simulation of the 20 degree single injector sector of the combustor has been performed. The LES approach allows for the simulation of the unsteady behavior of the flame. Validation studies showed a good agreement with the experimental data (Moin & Apte, 2000).

4. Full engine simulation

Here we present the results of an integrated multi-component simulation of a Pratt & Whitney aircraft engine. This simulation simultaneously computes the flow in the compressor, the combustor and the turbine, and each of the components exchanges flow data with its neighbors. The goal of this simulation is to demonstrate the ability to perform complex multi-physics multi-code simulation on a real world problem.

The domain consists of a 20 degree sector of the compressor, the combustor and the turbine. The geometries for each component are identical to the 20 degree sector component computations. The initial solution for the integrated simulation is provided by a combination of the component simulations.

4.1. *Operating conditions*

The operating conditions for the engine corresponds to cruise conditions. In the following, we define boundary conditions as true boundary conditions set by the problem, such as compressor inlet conditions, turbine outlet conditions and fuel inlet conditions. Interface conditions are those conditions specified at the interfaces between the components. As interface conditions, we have chosen a set of conditions usually used for gas turbine computations.

For the compressor portion, boundary conditions have to be specified at the inlet. Here, total temperature, total pressure and the flow directions are imposed. At the outlet of the compressor, the static pressure is imposed as an interface condition, which means the values are provided by the downstream flow solver CDP computing the combustor.

At the inlet the combustor receives the flow vector $[u, v, w]$ and uses an interface condition that we developed previously to convert RANS data into meaningful LES data (Schlüter *et al*, 2004). The fuel mass flow rate, defined as a boundary condition, corresponds to the cruise operating conditions. Although this is not necessary, the pressure
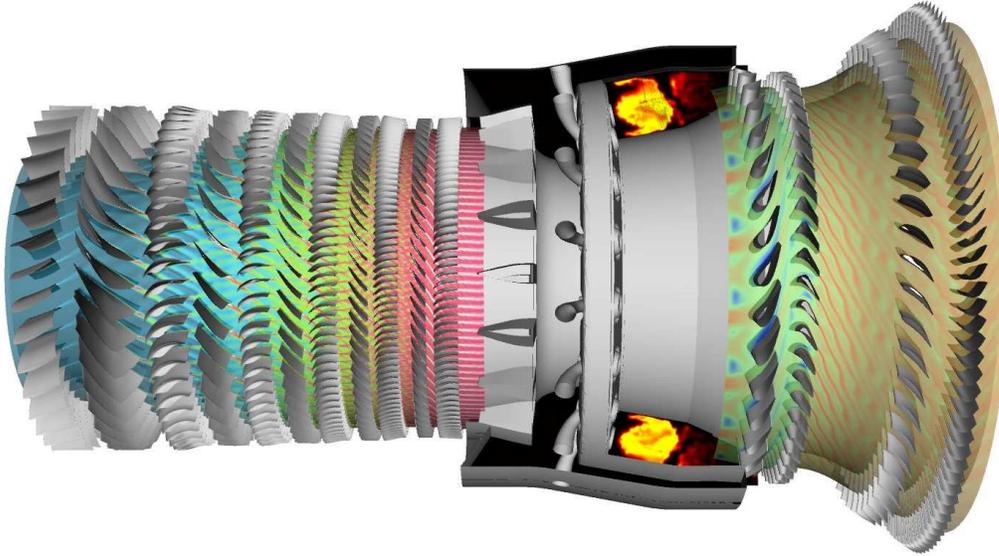
FIGURE 5. Simulation of the full high-spool. Compressor: axial momentum; Combustor: temperature; Turbine: axial momentum.

provided by CDP is fixed in the simulation shown below to simplify the coupling. However, simulations which are not shown here are being performed without this limitation. At the outlet of the combustion chamber, a body force technique is used to take the upstream effect of the turbine into account (Schlüter *et al*, 2005a). Here, the LES solver receives the flow velocities $[u, v, w]$ from the RANS turbine simulation for the overlap region of the computational domains of RANS and LES. The actual outlet of the combustor domain is far downstream in order to minimize the effect of the domain boundary and the convective outflow condition.

At the inlet the turbine receives the total pressure, the total temperature and the flow directions from the combustor. Since the initial solution of the combustor component has not reached steady state at the outlet, the temperature profile in the combustor does not correspond to the design conditions of the turbine. This is why we chose to use a constant temperature with radial variations at the turbine inlet. The temperature profile corresponds to the operating conditions of the turbine. Once the combustor simulation has reached a steady state, we will be able to couple the temperature as well. While this temperature profile in the current simulation is identical to the one used by the component simulation, the unsteady flow velocities delivered by CDP will create an unsteadiness at the inlet of the combustor. At the outlet of the turbine, we specify the static pressure as a boundary condition.

The communication between the components is handled by the coupling software CHIMPS. Since the turbomachinery meshes of each sector may not necessarily coincide with the sector mesh of the neighboring domain, the interface donor cells must be searched over the entire circumference of the engine. A fast search method has been developed to minimize the time spent on the sector searches. Vector components of exchanged flow variables are automatically rotated, dependent on the azimuthal offset of the neighboring domains.
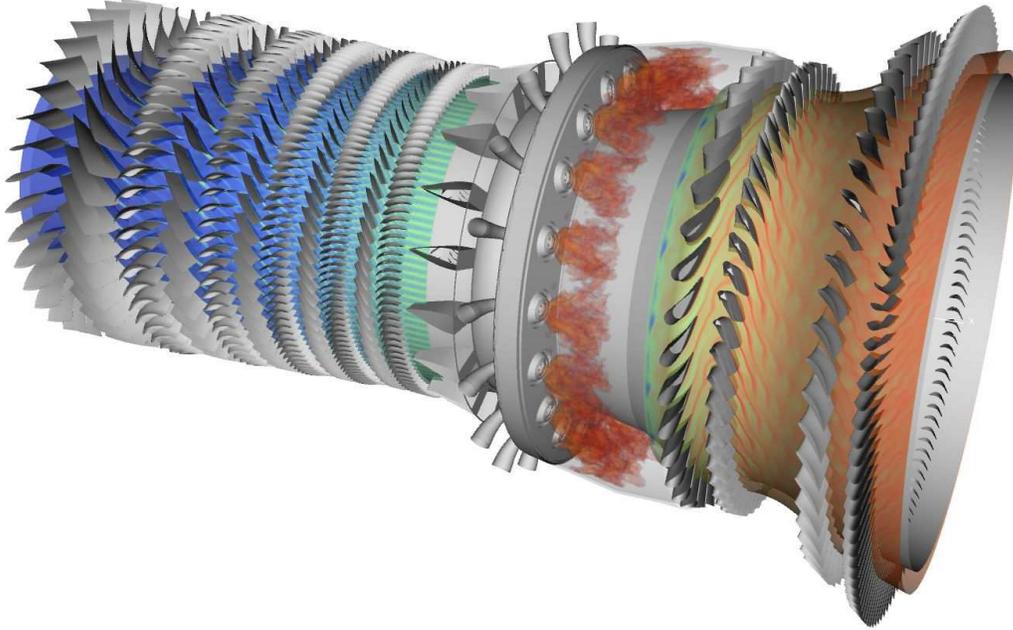
FIGURE 6. Simulation of the full high-spool. Compressor: entropy; Combustor: temperature; Turbine: entropy.

4.2. *Computational costs and results*

We performed this simulation on the Xeon Linux cluster at the US Department of Energy during a weekend. The simulation ran for 350 time-steps in 48 hours of wall clock-time on 502 processors. The load balancing between the solvers was determined by the component simulations. The compressor was run on 250 processors, the combustor on 96 processors, and the turbine on 156 processors. Since each of the solvers was run on a moderate number of processors, we did not use parallel I/O. The solvers required about 30-40% of their time in I/O. Since parallel I/O has recently been implemented in all of the solvers, we will be able to speed up the overall computation significantly. We estimate that a flow-through time of an entire high-spool of the engine can be computed within 14 days. The simulation is currently in production mode, and a detailed analysis will be made once a set of statistically significant data has been collected.

5. Conclusions

We have developed a new approach to simulate multi-component effects. In this approach, existing solvers are adapted for use in integrated simulations. We have developed a software module that allows the coupling of multiple solvers. The advantage of the module is that it is written in a general fashion, and solvers can be adapted easily to communicate with other solvers. The software module performs many of the required coupling tasks such as searches, interpolations, and process-to-process communication. The adaptation of solvers to the coupling module is facilitated, compared with previous approaches. We demonstrated this approach on a simulation of the entire high-spool of a Pratt & Whitney engine. The results are promising and we are able to show that such a simulation can be performed in a reasonable turnover time.

6. Acknowledgments

REFERENCES

ADAMIDIS, P., BECK, A., BECKER-LEMGAU, B., 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. *J. Mat. Proc. Tech.* **80**, 330–336.

ALONSO, J. J., MARTINELLI, L. AND JAMESON, A. 1995 Multigrid unsteady Navier-Stokes calculations with aeroelastic applications. *AIAA-95-0048.*

BATTEN, P., GOLDBERG, U., AND CHAKRAVARTHY, S. 2002 LNS-An approach towards embedded LES. *AIAA-2002-0427.*

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 Int'l 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-GT2003-38356.*

FERZIGER, J. H. 1996 Large eddy simulation: an introduction and perspective. In *New Tools in Turbulence Modeling*, Springer, 29–47.

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-91-1596.*

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

MOIN, P. AND APTE, S. 2004 Large-eddy simulation of realistic gas turbine combustors. *AIAA-2004-0330.*

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. In *IUTAM Symposium Turbulent Mixing Combustion*, Kingston, Canada.

RAMAN, V. AND PITSCH, H. 2005 Large-eddy simulation of a bluff-body stabilized flame using a recursive refinement procedure. *Combust. Flame.* **142** 329-347.

SAGAUT, P. 2002 *Large Eddy Simulation for Incompressible Flows*, second ed., Springer, Berlin.

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

SCHLÜTER, J., SHANKARAN, S., KIM, S., PITSCH, H. AND ALONSO, J. J. 2003b Towards multi-component analysis of gas turbines by CFD: Integration of RANS and LES flow solvers. *ASME-GT2003-38350.*

SCHLÜTER, J., PITSCH, H. AND MOIN, P. 2004 Large-eddy simulation inflow conditions for coupling with Reynolds-averaged flow solvers. *AIAA J.* **42**, 478–484.

SCHLÜTER, J., PITSCH, H. AND MOIN, P. 2005a Outflow conditions for integrated large-eddy simulation/Reynolds-averaged Navier-Stokes simulations, *AIAA J.* **43**, 156–164.

SCHLÜTER, J., WU, X., SHANKARAN, S., KIM, S., PITSCH, H. AND ALONSO, J. J. 2005b A framework for coupling Reynolds-averaged with large-eddy simulations for gas turbine applications. *ASME J. Fluids Eng.* **127**, 806–815.

SCHLÜTER, J., WU, X., WEIDE, E., HAHN, S., ALONSO,J.J. & PITSCH, H. 2005c Multi-code simulations: A generalized coupling approach. *AIAA-2005-4997*.

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

SPALART, P. R. 2000 Trends in Turbulence Treatments. *AIAA-2000-2306*.

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

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