45th AIAA Aerospace Sciences Meeting and Exhibit, January 8–11, 2007/Reno, NV

Integrated RANS/LES Computations of an Entire Gas Turbine Jet Engine

G. Medic,[*] G. Kalitzin,[†] D. You,[‡] E. v. d. Weide,[§] J. J. Alonso,[¶] and H. Pitsch[‖]

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

I. Introduction

The interaction between different components of a jet engine represents a very important aspect of the engine design process. Sudden mass flow-rate changes induced by flow separation and pressure waves, interaction of the unsteady wakes originating from the fan blades with the low-pressure compressor, high temperature streaks interacting with the first stages of the turbine are all complex unsteady phenomena that cannot be simply accounted for through boundary conditions of a single component simulation. Only simulations that integrate multiple engine components can describe these flow features accurately.

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 by both high Reynolds numbers and high Mach numbers. The 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 problem 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, removing all the details of the small scale turbulence; a turbulence model becomes necessary to represent the effects of turbulence on the mean flow.

The flow in the combustor, on the other hand, is characterized by multi-phase flow, intense mixing, and chemical reactions. The prediction of turbulent mixing 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.[1,2] 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,.[3] LES flow solvers have been shown in the past to be able to model simple flames and are currently being adapted for use in gas turbine combustors.[4,5]

In order to compute the flow in the entire jet engine, one needs to couple RANS and LES solvers. We have developed a software environment that allows a simulation of multi-component effects by executing multiple solvers simultaneously. Each of these solvers computes a portion of a given flow domain and exchanges flow data at the interfaces with its peer solvers (see Figure 1). The approach to couple two or more existing flow solvers has the distinct advantage of building upon the experience and validation that has been put

---

[*]Research Associate
[†]Research Associate
[‡]Research Associate
[§]Research Associate
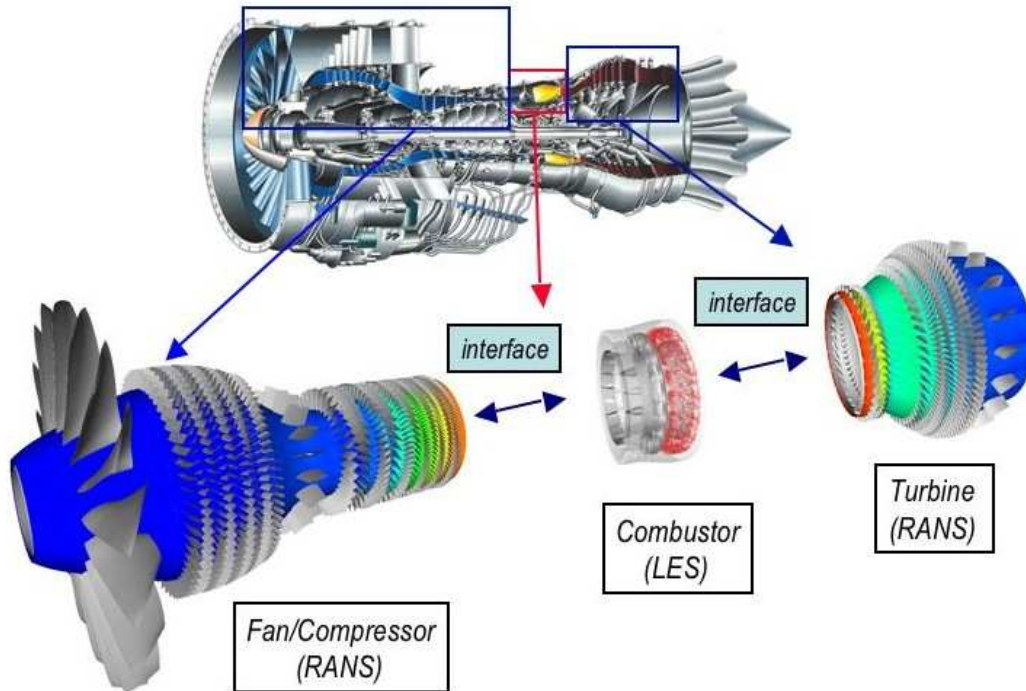[¶]Associate Professor
[‖]Assistant Professor

Figure 1. Decomposition of the engine for flow simulations.

into the individual codes during their development. It provides the possibility of running simulations in different domains at different time steps, and provides a higher degree of flexibility. We will demonstrate this approach in a simulation of a 20° sector of the entire gas turbine jet engine, encompassing the fan, low- and high-pressure compressor, combustor, high- and low-pressure turbine, and the exit nozzle. We will show that such a simulation can deliver important insight into the physics of interaction between different engine components within a manageable turnover time, which is necessary to be useful in the design process of an engine.

II.   Flow solvers

For the integrated computations presented here, we use flow solvers that are well-suited and tested for the individual components: a RANS flow solver for the turbomachinery parts and an LES flow solver for the combustor. These solvers were further adapted for efficient use on massively parallel platforms of up to several thousand CPUs, which are needed for these integrated computations of an entire jet engine.

A.   RANS flow solver

The RANS flow solver used in the computations 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.[6] 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. The turbulent viscosity is computed from a $k - \omega$ two-equation turbulence model and adaptive wall functions are employed to compute the boundary conditions. The dual time-stepping technique[7–9] is used for time-accurate simulations that account for the relative motion of moving parts as well as other sources of flow unsteadiness.

American Institute of Aeronautics and Astronautics

B. 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.[10] Here we summarize the main features of the methodology. The filtered Navier-Stokes equations are solved in an unstructured grid system using a Smagorinsky-type subgrid-scale (SGS) model.[11] The integration method used to solve the governing equations is based on a fully implicit fractional-step method. All terms, including cross-derivative diffusion terms, are advanced in time using the Crank-Nicholson method.

The Cartesian components of momentum, density, and pressure are stored at the nodes of the computational elements. Once density is obtained from a flamelet library, the continuity equation can be imposed as a constraint on the momentum field, with the time-derivative of density as a source term. This constraint is enforced by the pressure, in a manner analogous to the enforcement of the incompressibility constraint for constant density flows. The computational approach is to first advance the mixture fraction and the progress variable. The flamelet library yields the density, whose time-derivative is then computed. The momentum is predicted using the convective, viscous, and pressure-gradient at first. The predicted value of the momentum is then projected such that the continuity equation is satisfied.

C. Boundary conditions

The definition of the boundary conditions requires special attention, especially for LES. Because a part of the turbulent spectrum is resolved in the LES, the challenge is to regenerate and preserve the turbulence at the boundaries.

At the LES inflow boundary, the challenge is to prescribe transient turbulent velocity profiles from ensemble-averaged RANS data. Turbulent fluctuations at the inflow of the combustor have to be constructed using an additional LES computation. The fluctuations can be precomputed and stored in a database,[12] or computed on the fly from an auxiliary duct computation.[13]

For the RANS solver, inlet and exit boundary conditions are applied using the time-averaged solution from the LES.[14] Turbulence variables (such as k and ω) can also be computed on the fly from a quasi-2D RANS computation of an auxilliary duct at the turbine inflow.[13]
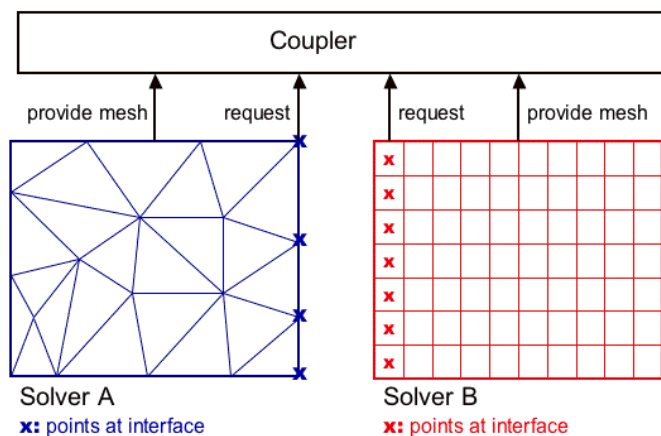

Figure 2. CHIMPS approach: solvers communicate location of their interface points and their mesh and solution to the coupler. The coupler determines how to provide information to the solver at the interface nodes.

III.   CHIMPS: Multi-solver coupling

Previous approaches to couple solvers were based on a pure MPI approach.[15–18] In that approach, MPI is used to let different solvers communicate directly with each other. The disadvantage of this approach is that the implementation is tedious and error prone since each MPI command in one solver requires a corresponding MPI command in the other solver. Furthermore, the search and interpolation routines have to be implemented in each solver separately.

American Institute of Aeronautics and Astronautics

A more effective approach consists in implementing all the coupling routines (communication, search, and interpolation) in a separate software module that performs these tasks: **C**oupler for **H**igh-Performance **I**ntegrated **M**ulti-**P**hysics **S**imulations (CHIMPS).[19] The solvers are now communicating with the coupler software only (Figure 2) and the coupler performs all searches and interpolations. In this latest version, the coupler supports both a script language such as Python and standard programming languages like Fortran90.

IV.   Full-engine simulations

In this section we present an integrated multi-component simulation of a Pratt & Whitney aircraft engine. This simulation simultaneously computes the flow in the fan/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° sector of all the components; in view of the full-engine simulation, this is the smallest sector that can be chosen since it contains one fuel injector. The initial solution for the integrated simulation is provided by a combination of the component simulations.

Note that 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.
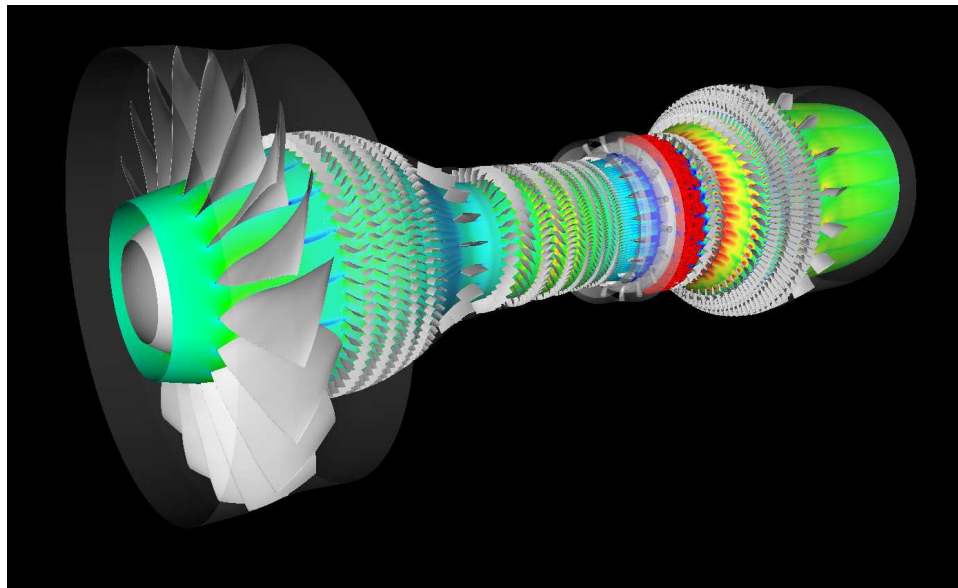

Figure 3.   Simulation of the entire engine: axial velocity.

A.   Operating conditions

The operating conditions for the engine correspond to cruise conditions; these define the boundary conditions for the engine: fan inlet conditions, turbine outlet conditions, and fuel inlet conditions. Boundary conditions are also specified at the interfaces, however here they are computed using the data from the neighboring component.

For the fan inlet, the total temperature, total pressure, and the flow directions are imposed. At the outlet of the compressor, the static pressure is imposed. The combustor receives at the inlet the flow vector $[u, v, w]$. The fuel mass flow rate is defined corresponding to the cruise operating conditions. The actual outlet of the combustor domain is far downstream in order to minimize the effect of the domain boundary

American Institute of Aeronautics and Astronautics

and the convective outflow condition. The turbine inlet receives the total pressure, the total temperature and the flow directions from the combustor; the quantities that are transfered are time-averaged on the fly as the computation proceeds. At the turbine outlet, we specify the static pressure.

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 are 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.

B.   Computational cost

The computational domain includes the fan, the low- and the high-pressure compressor, the combustor, the high- and low-pressure turbine, and the exit nozzle, as shown in Figure 3. We considered two sets of grids for the compressor: a finer grid consisting of approximately 57 million cells for the entire fan/compressor and a coarser grid consisting of approximately 8 million cells. The combustor grid contains 3 million cells and the fine grid for the turbine consists of approximately 15 million cells, whereas the coarser grid for the turbine consists of about 3 million cells. The time step has been chosen to assure that in the turbomachinery components we use at least 30 time steps for a blade passing in a blade row with the highest count and the highest rotational speed. This translates to about 11,500 time steps needed for a full wheel revolution of the slower low-pressure components and 3700 time steps for the faster rotating high-pressure components. In addition, estimates for the number of time steps needed for a flow-through time range from 10,000 time steps for the high-pressure spool core of the engine, to about 20,000 for the entire engine.

We have performed multiple simulations on a DOE ALC Xeon Linux cluster. The simulations typically run for 1500 time steps in 24 hours of wall-clock time on 700 processors, for the entire engine using the coarser grid for the fan/compressor and the turbine. The fan/compressor was run on 480 processors, the combustor on 80 processors, and the turbine on 140 processors. To obtain the same amount of time steps for the entire engine on the finer grid, approximately 4,000 processors are needed. A flow-through time for the entire engine can then be computed within 14 days of uninterrupted running.

An important component of these computations is the parallel I/O, which, depending on the desired frequency and extent of output data, can take up to 50 % of the run time (when saving output at every single time step). Here, we have chosen to save the output every 10 time steps.

C.   Results

First, the fidelity of the integrated simulation at 7,500 time-steps is examined by comparing the results at several axial locations (see Figure 4) to existing data provided by Pratt & Whitney. Circumferentially averaged radial profiles of total pressure and total temperature are shown in Figures 5 and 6, respectively. The results agree reasonably well with the data. However, the predictions are somewhat less accurate near the hub and the casing.

Next, we focus on the solution in the vicinity of the component interfaces and investigate three specific interaction phenomena. The first is the interaction of the wakes from the fan blades with the low-pressure compressor. The second concerns the influence of the wakes from the high-pressure compressor on the diffuser and the flow in the combustor, and the third one concerns the propagation of unsteady hot streaks from the combustor into the turbine.

The wakes originating from the fan blades propagate almost all the way through the low-pressure compressor, which effects the efficiency and flow capacity in the low-pressure compressor. This interaction between the fan and the low-pressure compressor is presented in detail in the companion paper.[20]

The axial velocity contours at the compressor/combustor interface plotted in the mid-span radial plane are shown in Figure 7. The wakes from the last row of vanes in the high pressure compressor are propagating into the diffuser, as these contours of the instantaneous axial velocity illustrate. We are currently examining the effect of these wakes on the stability (and possibly separation) of the flow in the diffuser, as well as its effect on the flow splits and the flow in the combustor chamber.

Figure 8 presents an isosurface of mean temperature in the combustor and turbine, indicating the high temperature streaks propagating through the combustor/turbine interface and into the turbine, i.e., the
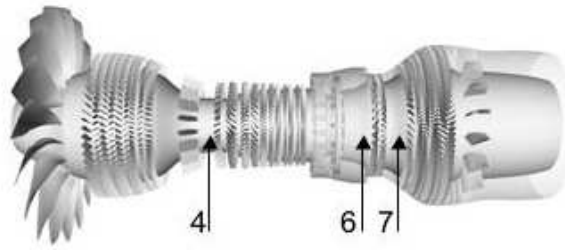
American Institute of Aeronautics and Astronautics
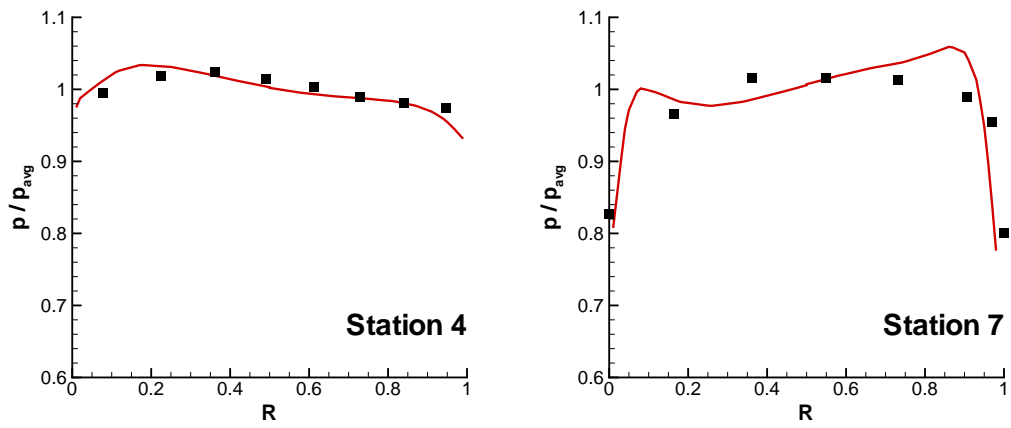
Figure 4.   Flow stations.

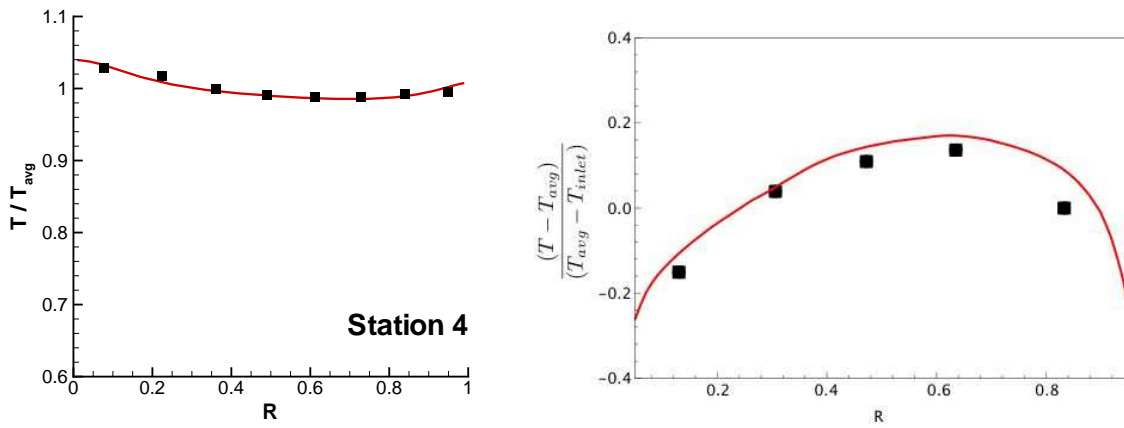**Figure 5.   Total pressure, circumferentially averaged profiles.**

Figure 6.   Total temperature, circumferentially averaged profiles.

time-averages of the flow variables from the combustor computation are passed to the turbine (total pressure, total temperature, and flow angles). At the turbine inflow there is a strong variation of temperature and axial velocity in the circumferential direction; we have observed a 10% circumferential variation of the temperature at the combustor/turbine interface at mid-span.
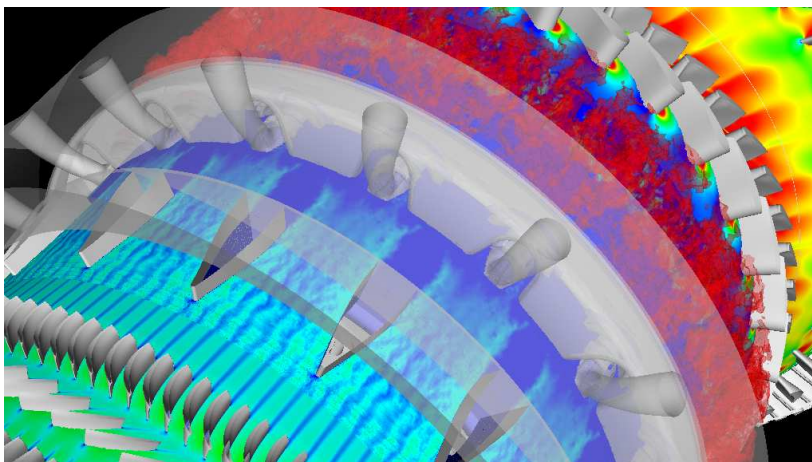

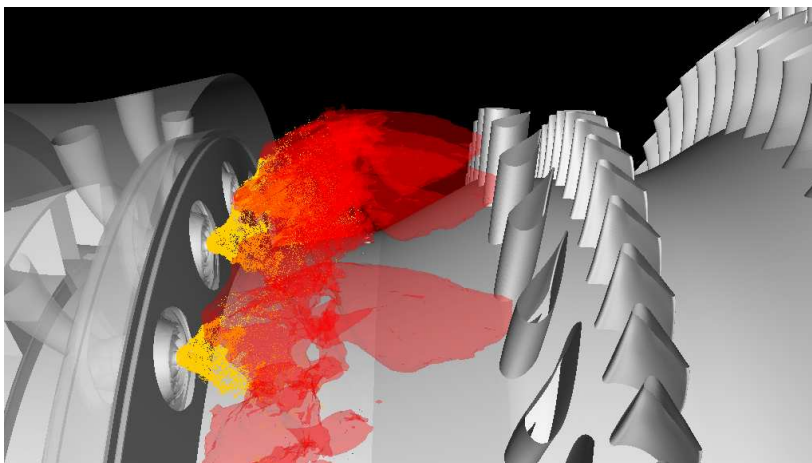**Figure 7. Compressor/combustor interface: axial velocity, mid-span.**

Figure 8. Combustor/turbine interface: isosurface of mean temperature.

V.   Conclusions

A new approach to simulate multi-component effects is proposed. In this approach, existing solvers are adapted for use in integrated simulations and a new software module has been developed to allow the coupling of multiple solvers. The advantage of using this module is that it is written in a general fashion and solvers can easily be adapted to communicate with other solvers. The software module performs many of the required coupling tasks, such as searches, interpolations, and process-to-process communication.

We demonstrated this approach in a simulation of the entire flow path of a Pratt & Whitney jet engine. The results are promising and we were able to show that the computational cost of such simulations is not prohibitive. The importance of interactions of the fan with the low-pressure compressor, the high-pressure compressor with the combustor-inlet diffuser, and the combustor with the high-pressure turbine are discussed. More details and a quantitative characterization of these interactions will be provided in future publications.

American Institute of Aeronautics and Astronautics

Acknowledgements

References

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

[2]Sagaut, P., *Large eddy simulation for incompressible flows*, Springer, Berlin, 2nd ed., 2002.

[3]Raman, V. and Pitsch, H., "Large-eddy simulation of a bluff-body stabilized flame using a recursive refinement procedure," *Comb. Flame*, Vol. 142, 2005, pp. 329–347.

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

[5]Constantinescu, G., Mahesh, K., Apte, S., Iaccarino, G., Ham, F., and Moin, P., "A new paradigm for simulation of turbulent combustion in realistic gas turbine combustors using LES," *ASME Turbo Expo 2003*, , No. GT2003-38356, 2003.

[6]Yao, J., Jameson, A., Alonso, J. J., and Liu, F., "Development and validation of a massively parallel flow solver for turbomachinery flows," *AIAA paper*, , No. AIAA-00-0882, 2000.

[7]Jameson, A., "Time Dependent Calculations Using Multigrid, with Applications to Unsteady Flows Past Airfoils and Wings," *AIAA paper*, , No. AIAA Paper 91-1596, 1991, AIAA 10th Computational Fluid Dynamics Conference, Honolulu, HI, June 1991.

[8]Alonso, J. J., Martinelli, L., and Jameson, A., "Multigrid Unsteady Navier-Stokes Calculations with Aeroelastic Applications," *AIAA Paper*, , No. AIAA 95-0048, 1995, AIAA 33rd Aerospace Sciences Meeting and Exhibit, Reno, NV, 1995.

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

[10]Mahesh, K., Constantinescu, G., Apte, S., Iaccarino, G., Ham, F., and Moin, P., "Large-eddy simulation of reacting turbulent flows in complex geometries,"' *Journal of Applied Mechanics*, Vol. 73, 2006, pp. 374-381.

[11]Germano, M., Piomelli, U., Moin, P., and Cabot, W., "A dynamic subgrid-scale eddy viscosity model," *Phys. Fluids*, Vol. A, No. 3 (7), July 1991, pp. 1760–1765.

[12]Schlüter, J. U., Pitsch, H., and Moin, P., "Large Eddy Simulation inflow conditions for coupling with Reynolds-averaged Flow Solvers," *AIAA Journal*, Vol. 42, No. 3, March 2004, pp. 478–484.

[13]Medic, G., You, D. and Kalitzin, G., 2006. "An approach for coupling RANS and LES in integrated computations of jet engines." CTR Annual Research Briefs.

[14]Kim, S., Schlüter, J., Wu, X., Alonso, J. J., and Pitsch, H., "Integrated simulations for multi-component analysis of gas turbines: RANS boundary conditions," *AIAA Paper*, , No. AIAA 2004-3415, July 2004.

[15]Shankaran, S., Liou, M.-F., Liu, N.-S., and anf J. J. Alonso, R. D., "A multi-code-coupling interface for combustor/turbomachinery simulations," *AIAA paper*, , No. AIAA 2001-0974, 2001.

[16]Schlüter, J. U., Shankaran, S., Kim, S., Pitsch, H., Alonso, J. J., and Moin, P., "Integration of RANS and LES flow solvers for simultaneous flow computations," *AIAA paper*, , No. AIAA 2003-0085, 2003.

[17]Schlüter, J. U., Shankaran, S., Kim, S., Pitsch, H., Alonso, J. J., and Moin, P., "Towards multi-component analysis of gas turbines by CFD: integration of RANS and LES flow solvers," *ASME paper*, , No. ASME GT2003-38350, 2003, ASME Turbo Expo 2003, June 16-19, 2003, Atlanta, GA.

[18]Schlüter, J. U., Wu, X., Kim, S., Alonso, J. J., and Pitsch, H., "A Framework for Coupling Reynolds-Averaged with Large Eddy Simulations for Gas Turbine Applications," *Journal of Fluids Engineering*, Vol. 127, No. 4, 2005, pp. 806–815, July 2005.

[19]Alonso, J.J,. et al., 2006. "CHIMPS: A high-performance scalable module for multi-physics simulations." *AIAA Paper*, AIAA-2006-5274.

[20]Kalitzin, G., Medic, G., van der Weide, E., and Alonso, J.J., "Interaction of Turbomachinery Components in Large-scale Unsteady Computations of Jet Engines," *AIAA Paper*, , No. AIAA 2007-0519, 1995, AIAA 45th Aerospace Sciences Meeting and Exhibit, Reno, NV, 2007.