Direct approach to the prediction of indirect combustion noise

By M. Kostka AND H. Pitsch

1. Motivation and Objective

While combustion is prevalent as energy conversion technology, environmental concerns pose restrictions on its use and demand a high degree of sophistication in the devices used for that purpose. Customers desire efficiency while governmental regulations impose limits on the amount of produced pollutants and noise. It is therefore necessary to incorporate these constraints into the design process. For the engineer of today this usually means that accurate and reliable ways to predict those quantities are needed. This brief communication describes efforts to develop an accurate model for the prediction of indirect combustion noise in realistic, confined geometries.

In the study of combustion noise, two mechanisms of noise generation can be distinguished, direct and indirect noise. Direct combustion noise results from the coupling of pressure fluctuations with the unsteady heat release in a flame. Indirect combustion noise results from the transport of entropy spots. It was identified as a relevant source of combustion noise by Marble & Candel (1977) and the authors provide an early analysis of the mechanism. Recent experimental investigations by Bake *et al.* (2005, 2007, 2009) confirm the significance of this mechanism in the process of combustion-generated noise. They provide some data for evaluation and assessment of computational approaches of higher fidelity than the one-dimensional analytical study mentioned. Part of their work has inspired very recent fundamental research in this area by Howe (2010).

While work on combustion noise has been carried out, (see for instance Ihme (2007) to name but one study) often some form of acoustic analogy has been used in an attempt to decouple acoustics from hydrodynamics. This method enables exploitation of the disparity between acoustic and convective time scales for low Mach number noise problems, as well as the option to use different numerical methods for the generally more challenging problem of noise propagation. These approaches are elegant, but cannot cope with confined geometries, which are the norm for technical devices. For this reason, an unstructured finite volume solver is developed to conduct compressible large eddy simulations of the experiment reported by Bake *et al.* (2005).

1.1. *General description*

In order to assess our approach to predicting combustion noise, we chose the experimental study conducted by researchers at DLR, Germany, as published in Bake *et al.* (2005, 2007, 2009). The experiment consists of two swirling, co-rotating air streams, in between which pure methane is injected. This generates a diffusion flame within a cylindrical combustion chamber, that is connected to an exhaust pipe via a converging-diverging nozzle to emulate gas turbine conditions, while maintaining diagnostic access. A thermocouple is placed into the combustion chamber as well as into the converging-diverging nozzle. Microphones record acoustic signals within the combustion chamber and the exhaust pipe.

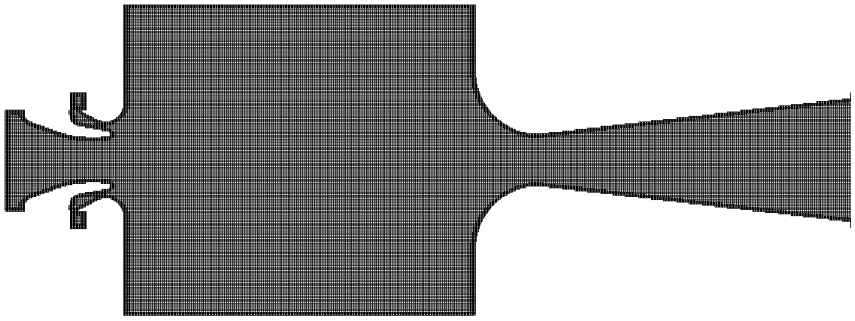

FIGURE 1. Slice through volume mesh

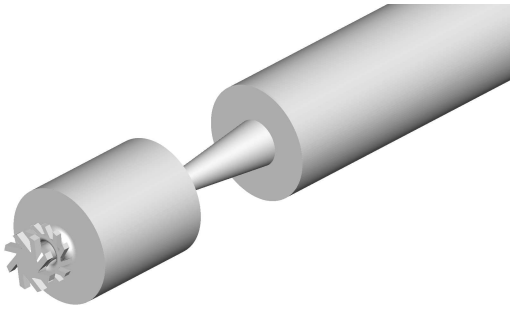

FIGURE 2. Surface description

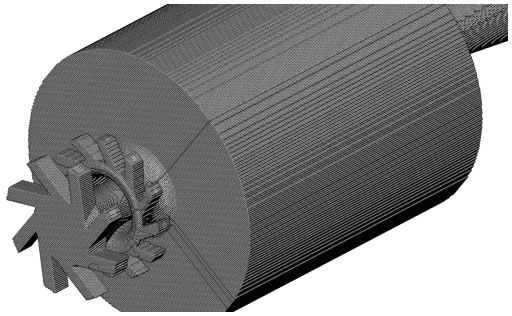

FIGURE 3. Volume mesh

From our communication with the experimentalists we have distinguished two separate experimental campaigns, which differ slightly in setup and conditions.

In the first case, corresponding to the work reported in Bake *et al.* (2005), a well-rounded nozzle was used to guide the outer, swirling air stream into the combustion chamber. The converging-diverging nozzle is of cross-sectional diameter 17 mm such as to yield, together with the set massflow rate for air and fuel (pure methane), $Ma = 0.17$.

In the more recent campaigns, the nozzle leading into the combustion chamber is replaced by a slightly different design such as to stabilize the flame within the chamber. Also, the cross-sectional diameter of the converging-diverging nozzle is narrowed to 7.5 mm, such that $Ma = 1$ is found in that nozzle.

Generating a high-quality mesh is of great importance, especially for inherently transient computations such as large eddy or direct numerical simulations.

From geometric models, we synthesized a closed-surface triangulation of the geometry, consisting of a number of STL files, each offering the possibility to prescribe distinct boundary conditions for the computations.

These triangulations, as shown in Figure 2, serve as input to our in-house meshing tool, which meshes the internal volume of the geometry with hexahedral cells, allowing for local refinement via hanging nodes.

2. Current calculations

Since we are interested in the prediction of combustion noise, we decided to focus on the case that excludes the additional challenges associated with sonic flow, such as the computational treatment of shock waves. This experiment was conducted at atmospheric

pressure such that the plenum conditioning the air before it enters the two swirlers was neither pressurized nor overheated. The massflow rate of air is 16.82 kg/h and 0.7232 kg/h methane is injected, which yields an equivalence ratio of about 0.73. The experimentalists provided us with a flow split for the air stream of

$$\frac{m_{inner}}{m_{outer}} = 0.75, \tag{2.1}$$

which we used to prescribe the massflow rates for both of the two swirlers.

2.1. *Combustion modeling*

In the context of large eddy simulation of combustion, an accurate, yet tractable model for the thermochemical conversion of reactants is important. For the present work, we used a variation of the so-called Flamelet-Progress variable approach put foward by Pierce & Moin (2004). It shares common ideas with models such as flamelet-generated manifolds, proposed by van Oijen *et al.* (2001), and flame prolongation of intrinsic low-dimensional manifolds, developed by Fiorina *et al.* (2003), in the sense that complex chemical processes involving dozens of species and hundreds of reactions are mapped in terms of a subset of scalars, which are solved as part of the flow solution procedure.

For the current model a transport equation for the conserved scalar Z and the progress variable C is being solved to locally estimate the chemical composition based on the flamelet model. Together with the total energy which is being solved for, the local temperature can then be computed. For the remainder of this report, we will refer to this model as the *'compressible FPVA'*, or short cFPVA, since, as outlined, it is based on the FPVA model introduced by Pierce & Moin (2004).

For the thermochemistry mapping, solutions of the steady flamelet equations

$$-\rho\frac{\chi}{2}\frac{\partial^2 Y_i}{\partial Z^2} = \omega_i \tag{2.2}$$

provide a function $Y_i = F(Z, \chi)$, where Y_i represents species mass fractions. Given an appropriate definition of a progress variable, C, this functional dependence can be mapped as: $Y_i = F(Z, \chi) \rightarrow F'(Z, C)$.

This mapping is stored as a look-up table and provides thermochemical quantities as a function of the local values for Z and C. We would like to stress that this approach allows for compressibility and viscous heating effects in the flow computation. However, the chemistry itself is still based on the low Mach number assumption, since the species mass fractions only depend on the transported scalars $\widetilde{Z}$ and $\widetilde{C}$, as well as the subgrid scale variance of Z, $\widetilde{Z''^2}$, the tilde denoting filtered quantities as usual within the LES context. Therefore, the approach in the present form is not able to capture direct noise, since it does not provide a dependence of reaction rates on pressure. A straightforward way of alleviating this problem, however, is to extend the dimensions of the pre-computed chemistry table with pressure.

2.2. *Numerical issues*

Much effort has gone into the development of a numerical method or strategy that is able to handle both the issues regarding compressibility and multi-material mixing as well as the strong heat release due to combustion. A hybrid WENO-Central method has been developed that seems to offer good shock-capturing properties, while, at the same time, preserving small scales of turbulence away from such discontinuities.

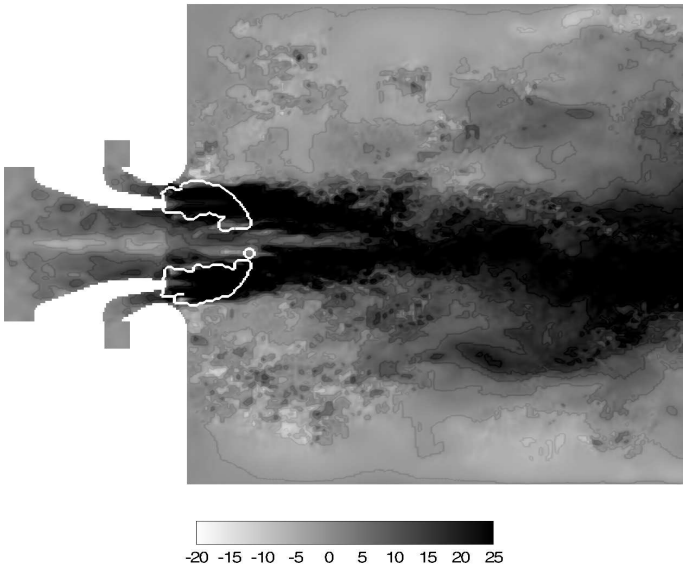

FIGURE 4. Region treated by first-order operators shown as white line, overlaid onto instantaneous velocity field. Velocity given in m/s.

For the DLR combustion noise experiment configuration, even a pure WENO treatment was not able to handle the strong gradients close to the nozzle, where methane enters the combustion chamber. Local refinement alleviated this problem somewhat, but considering the maximum Mach number experienced in this experiment and the acoustic CFL constraint our explicit solver has to obey, timestep size became impractically small.

The remedy, for the time being, is to treat a small region around the fuel nozzle by stable first-order operators, ensuring robustness of the computation, while throughout the rest of the domain a WENO scheme is used. In Figure 4, the region enclosed by the white line is treated by first-order operators. Within a given sphere around the nozzle, control volumes with a value of mixture fraction exceeding a certain threshold are treated in a first-order manner, so that this practice is limited to as small a region as possible and applied only where the flame is located. Given this strategy and current mesh resolution, a reasonable timestep size could be achieved.

2.3. *Initialization*

In variable density computations, using a Poisson equation to enforce continuity, it is rather straightforward to efficiently initialize a reactive calculation. Usually a mixing case is run. At some point, thermodynamic conditions as well as the density field are adjusted to a burning state, and a few flow-through times wash out unphysical, transient artifacts.

Present experience for compressible computations, however, prohibits such practice. Various adjustments of this strategy have been tried, but none has proved practical. Usually strong pressure waves are emitted from the flame, and the flow field changes so drastically that it is most efficient to just start a reactive computation from scratch, ensuring a burning flame from the start, rather than attempting to capture a pseudo ignition process or attempting an ad hoc ignition.

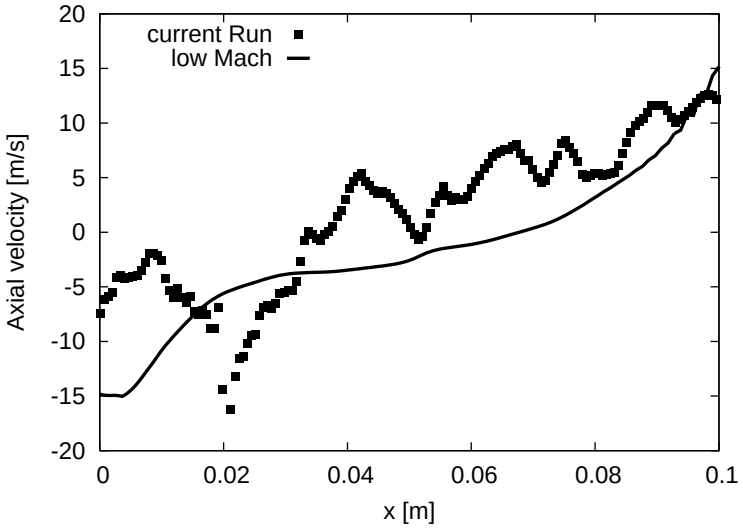

FIGURE 5. Instantaneous profile of axial velocity field inside combustion chamber. Overlaid onto the mean obtained from a benchmark low Mach number calculation.

Similarly, attempts have been made to artificially raise the Mach number while maintaining nominal Reynolds and Damköhler numbers to facilitate rapid initialization of the flow field, with the hope that a rescaling to nominal Mach number would be possible for the actual calculation.

Again, however, rescaling proved to alter the flow field so drastically that this practice has been abandoned.

3. Results

Currently, we are validating the computations with regards to benchmark runs conducted with a solver using a low Mach number approximation for large eddy simulation, which has been described in Desjardins *et al.* (2008).

Figure 5 shows the instantaneous profile of axial velocity, in comparison with a mean profile from the benchmark solution. Results look promising, although it has to be borne in mind that this is still work in progress.

Currently, we are working on the generalization of the three-dimensional Navier-Stokes characteristic boundary condition (3D-NSCBC) methodology by Lodato *et al.* (2008), for the unstructured solver for a reactive system. This boundary treatment builds upon earlier work by Poinsot & Lele (1992) and Yoo & Im (2007).

It has been observed that acoustic waves trapped inside the domain cause unphysical fluctuations in the mass flow rates, which is undesirable. We hope to use the 3D-NSCBC treatment to improve the physical accuracy of the boundary condition, reduce numerical reflection and improve the induced mass flow rates.

4. Future Work

Once validation of the 3D-NSCBC implementation has been carried out, it will be used to assess the quantitive behavior and the benefits it provides. Validation of the

computation will be preformed with respect to experimental velocity data as well as to the data for sound pressure levels obtained from a microphone in the exhaust duct.

5. Acknowledgments

This work is funded by NASA and their financial support is gratefully acknowledged. Furthermore, the help of the experimental and computational groups at DLR in Berlin and Stuttgart is acknowledged with great appreciation. Many thanks go to Dr. Guido Lodato for his work and help regarding the implementation and port of the 3D-NSCBC method.

REFERENCES

Bake, F., Kings, N., Fischer, A. & Röhle, I. 2009 Experimental investigation of the entropy noise mechanism in experimental investigation of the entropy noise mechanism in experimental investigation of the entropy noise mechanism in aero-engines. *International Journal of Aeroacoustics* **8** (1&2), 125–142.

Bake, F., Michel, U. & Roehle, I. 2007 Investigation of entropy noise in aero-engine combustors. *Journal of Engineering for Gas Turbines and Power Generation* **129**, 370–376.

Bake, F., Michel, U., Röhle, I., Richter, C., Thiele, F., Liu, M. & Noll, B. 2005 Indirect combustion noise generation in gas turbines. *11th AIAA/CEAS Aeroacoustics Conference, Monterey, California* .

Desjardins, O., Blanquart, G., Balarac, G. & Pitsch, H. 2008 High order conservative finite difference scheme for variable density low Mach number turbulent flows. *Journal of Computational Physics* **227**, 7125–7159.

Fiorina, B., Baron, R., Gicquel, O., Thevenin, D., Carpentier, S. & Darabiha, N. 2003 Modelling non-adiabatic partially premixed flames using flame-prolongation of ILDM. *Combustion Theory Modelling* **7** (3), 449–470.

Howe, M. 2010 Indirect combustion noise. *Journal of Fluid Mechanics* **659**, 267–288.

Ihme, M. 2007 Pollutant formation and noise emission in turbulent non-premixed flames. PhD thesis, Stanford University.

Lodato, G., Domingo, P. & Vervisch, L. 2008 Three-dimensional boundary conditions for direct and large-eddy simulation of compressible viscous flows. *Journal of Computational Physics* **227**, 5105–5143.

Marble, F. & Candel, S. 1977 Acoustic disturbance from gas non-uniformities convected through a nozzle. *Journal of Sound and Vibration* **55** (2), 225–243.

van Oijen, J., Lammers, F. & de Goey, L. 2001 Modeling of complex premixed burner systems by using flamelet-generated manifolds. *Combustion and Flame* **127**, 2124–2134.

Pierce, C. & Moin, P. 2004 Progress-variable approach for large-eddy simulation of non-premixed turbulent combustion. *Journal of Fluid Mechanics* **504**, 73–97.

Poinsot, T. & Lele, S. 1992 Boundary conditions for direct simulations of compressible viscous flows. *Journal of Computational Physics* **101**, 104–129.

Yoo, C. & Im, H. 2007 Characteristic boundary conditions for simulations of compressible reacting flows with multi-dimensional, viscous and reaction effects. *Combustion Theory and Modelling* **11** (2), 259–286.