LARGE-EDDY SIMULATION OF TURBULENT SUPERSONIC COLD FLOW IN RAMP-CAVITY COMBUSTOR WITH INJECTOR

Zia Ghiasi[1], Dongru Li[1], Jonathan Komperda[1], Farzad Mashayek[1], Arnab Chaudhuri[2] & Gustaaf B. Jacobs[2]
[1]*Department of Mechanical and Industrial Engineering, University of Illinois at Chicago, Chicago, USA*
[2]*Department of Aerospace Engineering, San Diego State University, San Diego, USA*

Abstract Development of efficient supersonic propulsion systems requires a deep understanding of the flow within such combustors. Performing numerical simulation of flow in a scramjet engine is challenging due to the concurrent presence of shock waves and turbulence. In this work, we employ a high-order discontinuous spectral element method (DSEM). An entropy-based artificial viscosity method is used to capture the shock and the turbulence model is the standard Smagorinsky-Lilly model in conduction with two new sensors that prevent addition of turbulent viscosity in non-turbulent regions. Results are presented for the simulation of supersonic turbulent cold flow in a ramp-cavity combustor with an injector at the ramped side of the cavity. The Reynolds number based on the cavity height and mean inlet flow velocity is $Re = 25,288$ and the Mach number is $Ma = 2$.

INTRODUCTION

We employ a high-order numerical method to simulate supersonic turbulent flows. The method is capable of locating strong shocks and capturing them within the numerical stencil as well as resolving turbulence. The main challenge in supersonic turbulent flow simulations arises from the contradictory properties of numerical methods designed to treat shocks and turbulence where the numerical scheme needs to satisfy two competing requirements. These requirements are capturing different types of discontinuities and simultaneously resolving the broadband scales of turbulence. The shock capturing is usually achieved through adding dissipation to the flow at the shocks in order to smear them sufficiently such that they can be represented on the numerical stencil. However, the use of a shock-capturing scheme can crucially affect the fidelity of the solution. A poorly designed shock capturing method can be over-dissipative and smear the solution, excessively dissipate turbulence, and thus lead to an inaccurate representation of the flow discontinuities and turbulence. One of the objectives of the present research is to design an effective and accurate shock capturing method for high-order numerical methods.

FORMULATION AND METHODOLOGY

In this work, a nodal collocation form of the discontinuous spectral element method (DSEM) [1] is used as the flow solver. DSEM approximates the solution of the conserved variables on a high-order local basis function inside non-overlapping elements that may be oriented arbitrarily within an unstructured grid. DSEM has negligible diffusion and dispersion errors and is spectrally convergent for smooth solutions. The governing equations are the non-dimensional equations for conservation of mass, momentum, and energy, the so-called Navier-Stokes equations, in three-dimensional Cartesian coordinates.

The turbulence model is the standard Smagorinsky-Lilly turbulence model. The subgrid model introduces a turbulent viscosity, which is approximated as

$$\nu_t = (C_s \Delta_G)^2 \sqrt{2 \overline{S}_{ij} \overline{S}_{ij}}, \tag{1}$$

where C_s is the Smagorinsky constant; $\overline{S}_{ij}$ is the filtered rate of strain, and Δ_G is a length scale, representing the average distance between solution points in each element. In addition, we developed two new sensors that are multiplied by the turbulence viscosity. The first sensor adds the turbulent viscosity only to the areas where the turbulence is strong. The value of this sensor is zero where there is no density gradient (regions with no turbulence) and approaches 1 where the magnitude of the density gradient is high (regions with strong turbulence). The second sensor removes the undesired viscosity from the shocked areas. The value of this switch varies from almost 0, for shock regions, to 1, for incompressible regions. The idea is that in the high turbulence areas the value of the vorticity is high, whereas in the shocked areas the dilatation is strong.

The shock capturing method used in this work is a modified version of the entropy viscosity method [2] for supersonic turbulent flows proposed by Abbassi *et al.* [3] The entropy transport equation for Navier-Stokes system of equations reads

$$\frac{\partial s}{\partial t} + \frac{\partial}{\partial x_j}(u_j\, s) - \frac{\Phi + \Gamma}{T} - \Lambda \geq 0\,, \tag{2}$$

with s denoting the fluid's entropy. In Eq. (2), Φ and Γ are the entropy generation terms due to viscous dissipation and heat conduction within the flow, respectively. A residual is defined based on this entropy transport equation and used to both identify discontinuities in the flow and define the magnitude of the artificial viscosity in those regions. The artificial viscosity is also multiplied by the Ducros switch [4] to avoid adding viscosity in shock-free turbulent areas.

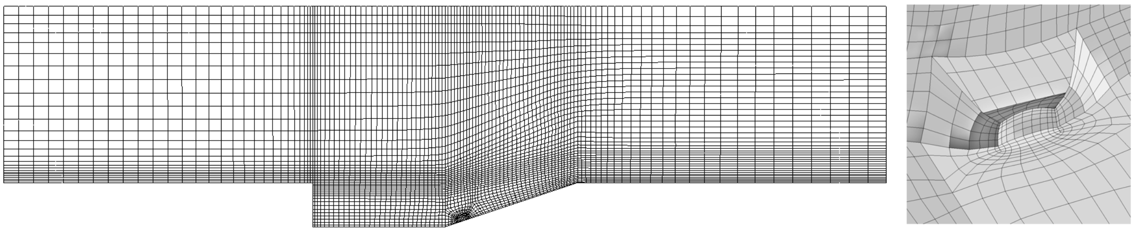

Figure 1. Left: The computational domain. Right: Locally refined grid for the injector.

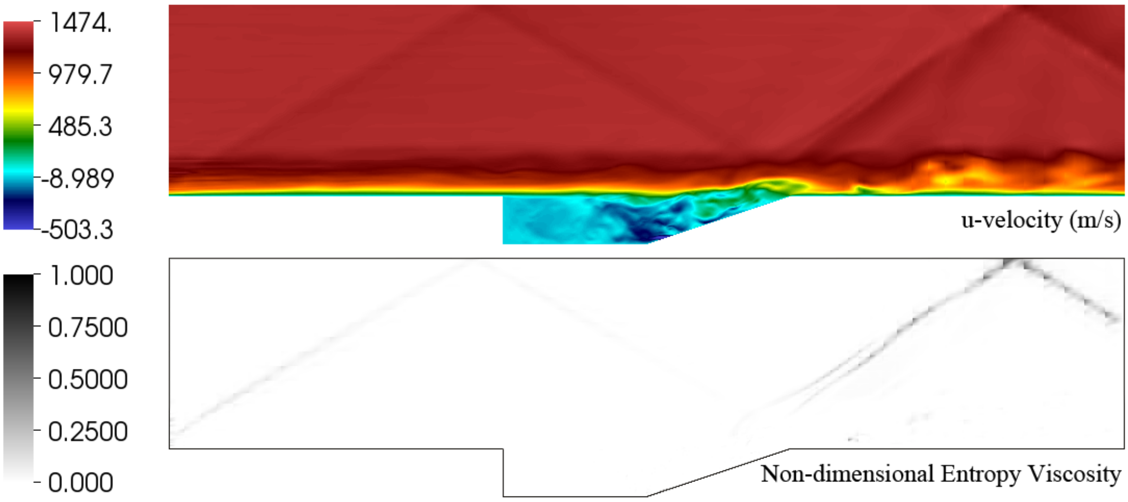

Figure 2. Instantaneous streamwise u-velocity (m/s) and non-dimensional entropy viscosity after 20 flow-through times.

RESULTS

A simulation has been conducted for the supersonic turbulent cold flow ($Re = 25,288$, $Ma = 2$) in a ramp-cavity combustor with an injector at the ramped side of the cavity. The computational grid used for this simulation is shown in Fig. 1. The grid consists of $70,116$ elements with polynomial order of $P = 2$ resulting in 1,893,132 solution points. DSEM allows to efficiently refine the grid in the vicinity of the injector in order to resolve the small scale mixing effects of the injector. The local refinement of the unstructured grid is depicted in Fig. 1 (right).

At the inlet, air enters the combustor at $Ma = 2$, atmospheric pressure, and temperature of 800 K. The injector is placed horizontally and blows air in the counter-streamwise direction. Figure 2 shows the instantaneous contours of u-velocity and the non-dimensional entropy viscosity after 20 flow-through times, plotted in a plane perpendicular to the spanwise direction, passing through the center of the injector. The flow-through time is calculated based on the mean velocity at the inlet and the length of the domain in the streamwise direction. The entropy viscosity is scaled by the dynamic viscosity of the fluid. The shock capturing method accurately captures the shock and almost no entropy viscosity is added to the turbulent areas. Various moments of the flow are calculated and will be presented along with those for a case without the injector to study the effect of the injector.

References

[1] D. Kopriva. A Staggered-grid Multidomain Spectral Method for Compressible Navier-Stokes Equations. *J. Comput. Phys.* **143**: 125–158, 1998.

[2] J.-L. Guermond, R. Pasquetti, and B. Popov. Entropy Viscosity Method for Nonlinear Conservation Laws. *J. Comput. Phys.*, **230** (11): 4248–4267, 2011.

[3] H. Abbassi, F. Mashayek, and G. Jacobs. Entropy Viscosity Approach for Compressible Turbulent Simulations using Discontinuous Spectral Element Method. *AIAA Paper#* 0947, 2014.

[4] F. Ducros, V. Ferrand, F. Nicoud, C. Weber, D. Darracq,C. Gacherieu, and T. Poinsot. Large-Eddy Simulation of the Shock/Turbulence Interaction. *J. Comput. Phys.* **152** (2): 517–549, 1999.