On the Development of Large Eddy Simulation Tools for Compressible Jet Flow Configurations

Carlos Junqueira-Junior*,1, Sami Yamouni*,2 and João Luiz F. Azevedo†

* Instituto Tecnológico de Aeronáutica
São José dos Campos - Brazil
[1] junior.hmg@gmail.com [2] sami.yamouni@gmail.com

† Instituto de Aeronáutica e Espaço
São José dos Campos - Brazil
joaoluiz.azevedo@gmail.com

ABSTRACT

One of the main design issues related to launch vehicles lies on noise emission originated by complex interaction between high temperature/high velocity exhaustion gases and the atmospheric air. These emissions, which have high noise levels, can damage the launching structure or even be reflected upon very delicate structures at the top of the vehicle. Moreover, the resulting pressure fluctuations can damage the structure or equipment at different parts of the launcher by vibrational acoustic stress. These acoustics design constraints have encouraged the studies of aeroacoustic fields around compressible jet flows. Instituto de Aeronáutica e Espaço (IAE) in Brazil is interested in such flow conditions for rocket design applications. Therefore, the current work addresses the numerical study of unsteady turbulent compressible flows for aeroacoustic applications at IAE. A novel large eddy simulation tool is developed in order to reproduce high fidelity results of compressible jet flows which are used for aeroacoustic studies using the Ffowcs Williams and Hawkingg approach. The numerical solver is an upgrade of a Reynolds-averaged Navier-Stokes code developed by the research group. Generally, large eddy simulations demand very dense grids. Hence, high performance computing is a requirement for such simulations. Therefore, the original framework is rewritten in Fortran 90 standards, dynamic memory allocation and parallel computation have been added to the code. The communication between processors is performed by message passing interface protocols (MPI). The CFD general notation system (CGNS) [1] is also included in the numerical tool. The large eddy simulation formulation is written using the finite difference approach. Inviscid numerical fluxes are calculated using a second-order accurate centered scheme with the explicit addition of artificial dissipation. A five-step second-order accurate Runge-Kutta scheme is the chosen time marching method. System I formulation [2] is applied in order to model the energy equation of the filtered Navier-Stokes equations. The classical Smagorinsky model [3] is the chosen turbulence closure for the present work. A numerical simulation of a perfectly expanded jet with Mach number equal to 1.37 and Reynolds number of the jet equal to 5×10^5 is performed and presented here. Time averaged results are compared with numerical solutions of Mendez [4], Bodony [5] and experimental data of Lau [6]. Moreover, the speedup and the computational performance of the code are evaluated and discussed.

REFERENCES

[1] Poirier, D.; Allmaras, S. R.; McCarthy, D. R.; Smith, M. F. and Enomoto, F. Y., “The CGNS System”, AIAA Paper No. 98-3007, 1998.

[2] Vreman, A. W., *“Direct and Large-Eddy Simulation of the Compressible Turbulent Mixing Layer”,* Ph.D. Thesis, Universiteit Twente, 1995.

[3] Smagorinsky, J., “General Circulation Experiments with the Primitive Equations: I The Basic Experiment”, *Monthly Weather Review,* Vol. 91, No. 3, March 1963, pp. 99-164.

[4] Mendez, S.; Shoeybi, M.; Sharma, A.; Ham, F. E. and Moin, S. K. L. P., “Large-Eddy Simulation of Perfectly-Expanded Supersonic Jets: Quality Assessment and Validation”, AIAA Paper No. 2010-0271, 2010.

[5] Bodony, D. and Lele, S. K., “On Using Large-Eddy Simulation for the Prediction of Noise from Cold and Heated Turbulent Jets”, *Physics of Fluids,* Vol. 17, No. 8, Aug. 2005.

[6] Lau, J. C.; Morris, P. J. and Fisher, M. J., “Measurements in Subsonic and Supersonic Free Jets Using a Laser Velocimeter”, *Journal of Fluid Mechanics,* Vol. 93, No. 1, 1979, pp. 1-27.