Materials Research Proceedings 37 (2023) 222-225 https://doi.org/10.21741/9781644902813-48

Development of a DNS solver for compressible flows in generalized curvilinear coordinates

Giulio Soldati[1,a *], Alessandro Ceci[1,b] and Sergio Pirozzoli[1,c]

[1] Via Eudossiana 18, Rome 00184, Italy

[a]giulio.soldati@uniroma1.it, [b]alessandro.ceci@uniroma1.it, [c]sergio.pirozzoli@uniroma1.it

Keywords: Generalized Curvilinear Coordinates, Shock-Wave/Boundary Layer Interaction, Turbulent Compression Ramp, Curved Channel Flow

Abstract. We present a solver for DNS of turbulent compressible flows over arbitrary shaped geometries. The code solves the compressible Navier-Stokes equations in a generalized curvilinear coordinates system, using high-order central finite-difference schemes combined with WENO reconstruction for shock-waves treatment. An innovative stabilization strategy for central schemes based on skew-symmetric-like splitting of convective derivatives is used. The code is oriented to modern HPC platforms thanks to MPI parallelization and the ability to run on GPU architectures. The robustness and accuracy of the present code is assessed both in the low-subsonic case and in the supersonic case. We show here the results of a turbulent curved channel flow and a turbulent supersonic compression ramp, which proved to be in excellent match with previous studies.

Introduction

Compressible flows over curved surfaces are ubiquitous in the aerospace field, both in external (e.g., aircraft wings) and internal (e.g., turbomachinery) configurations. The dynamics of these flows involves complex phenomena, such as shock-wave boundary layer interactions, which still need further investigation, and make it challenging to accurately estimate skin friction and wall heat transfer. A key tool in this context is the DNS, which demands high-order numerical schemes and good shock-capturing capabilities to accurately describe high-speed flows. However, due to the numerical complications arising from geometric complexities, the application of the DNS is often limited to simple Cartesian cases [1,2]. Dealing with complex geometries leads to the use of local mesh refinement or unstructured meshes, which in turns causes high computational cost or decreased accuracy, respectively [3]. Another approach to deal with such geometries is the use of body fitted mesh in a generalized curvilinear coordinates system, which guarantee an accurate simulation of the fluid dynamics near the wall at a competitive computational cost. In this work, we developed a solver for DNS of turbulent compressible flows over complex geometries, using high-order central finite-difference schemes in a generalized curvilinear coordinate system. Central approximations of the Navier-Stokes equations, being non-dissipative, exhibit numerical instability when used at small viscosity. To address this problem, we implemented energy-preserving schemes [4], allowing to accurately simulate the wide range of turbulent scales without relying on artificial diffusivity [5] or filtering of physical variables [6]. Moreover, those schemes can be efficiently combined with modern shock-capturing methods as WENO reconstructions, yielding hybrid schemes that currently represent an optimal strategy for the computation of shocked flows [7].

Methodology

The code solves the compressible Navier-Stokes equations for a perfect gas in a generalized curvilinear coordinate system:

$$\frac{1}{J}\frac{\partial \mathbf{Q}}{\partial t}+\frac{\partial \mathbf{F}_j}{\partial \xi_j}=\frac{\partial \mathbf{F}_j^v}{\partial \xi_j}$$

where $\mathbf{Q}$ is the vector of conservative variables, $\mathbf{F}_j$ is the vector of convective fluxes, $\mathbf{F}^v_j$ is the vector of viscous fluxes and J is the Jacobian of the coordinate transformation. Here, we consider stationary grids. A full description of the fluxes vectors can be found in [8]. A curvilinear body-fitted mesh is first represented in the physical space, x_i. Through the transformation $x_i(\xi_j)$ it is then mapped to the computational space, ξ_j, where it can be seen as a regular hexahedron. Non-uniform skewed input cells of the mesh are thus re-stretched into uniform cubical cells. Finite-difference schemes are applied in the computational space to approximate spatial derivatives, which must be reconstructed in the physical space by using the metrics, $\partial\xi_j/\partial x_i$. Since the mesh is directly described in the physical space, first we compute the inverse of the metrics, $\partial x_i/\partial\xi_j$, by numerically deriving the physical mesh coordinates; then the metrics are obtained with a matrix inversion. To guarantee free-stream preservation, we use the same approximation for both metric and convective derivatives [9].

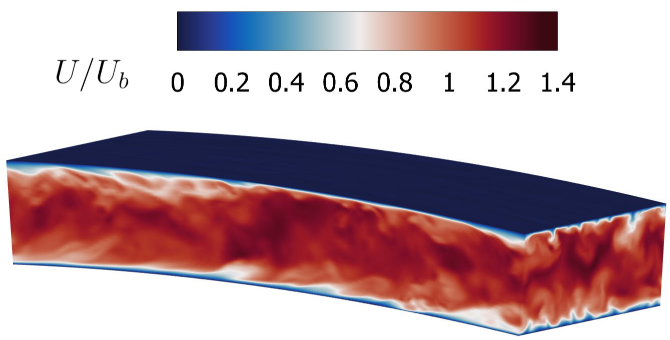
In smooth regions of the flow, a skew-symmetric-like splitting of the convective derivatives is employed. This approach guarantees preservation of kinetic energy in the semi-discrete, inviscid low-Mach-number limit. A computationally effective implementation of convective derivatives cast in split form was proposed by Pirozzoli [10]. The locally conservative formulation allows straightforward hybridization of central schemes with classical shock-capturing reconstructions. In our case, shock-capturing capabilities rely on WENO reconstruction of the numerical flux in the proximity of discontinuities. To judge on the local smoothness of the numerical solution and switch between the energy preserving and the shock capturing discretization, our code relies on a modified version of the Ducros shock sensor [11]. As for the viscous terms, they are expanded to Laplacian form to avoid odd-even decoupling phenomena. Spatial derivatives are approximated in the computational space with central formulas and reconstructed in the physical space by applying the chain-rule. The accuracy order of each scheme can be selected by the user and goes up to eight in the case of central schemes, up to seventh for WENO ones. The system is advanced in time using a three-stage, third order Runge-Kutta scheme [12].

The code is designed to efficiently work on the most common HPC architectures operating today. MPI parallelization and CUDA-Fortran porting enable the use of multi-GPUs architectures, while retaining the possibility to compile and use the code on standard CPU-based systems.

Results

We simulated low-Reynolds-number, mildly curved, turbulent channel flow. The computational domain is bounded by sectors of concentric cylinders, with a centreline radius of curvature r_c = 79h, where h is the channel semi-height. An imposed mean-pressure gradient in the azimuthal direction drives the flow. Periodic boundary conditions are imposed in the streamwise and spanwise directions, so that the simulated flow is fully evolved. The resolution is 216x72x144 in the streamwise, wall-normal and spanwise directions, respectively. Mach number and Reynolds number based on the bulk velocity are set to $M_b = 0.1$ and $Re_b = 2600$. The Reynolds numbers based on the inner and outer friction velocities resulted in $Re^i_\tau = 156$ and $Re^o_\tau = 178$, respectively. Fig. 1 illustrates an instantaneous field of the velocity magnitude U normalized with respect to bulk velocity U_b. Fig. 2 shows the root-mean-square (rms) of the velocity fluctuations of the present DNS (STREAmS) together with the data from Moser and Moin [13] and Brethouwer [14]. The excellent agreement of the results can be noted.

Materials Research Proceedings 37 (2023) 222-225 https://doi.org/10.21741/9781644902813-48



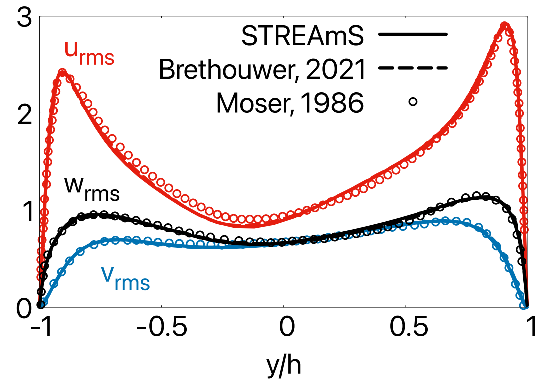
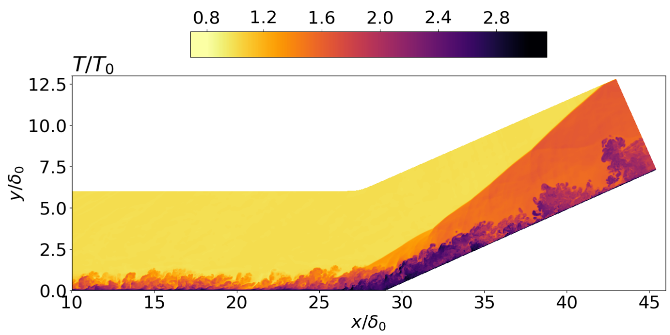


Fig. 1 Instantaneous field of the velocity magnitude.

Fig. 2 Rms of the velocity fluctuations.



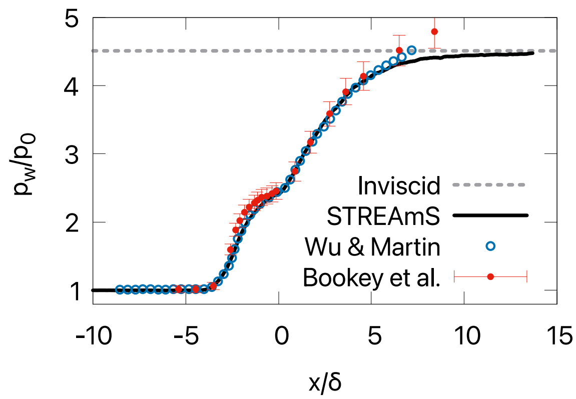


Fig. 3 Instantaneous field of the temperature.

Fig. 4 Mean wall-pressure distribution.

Shock wave/turbulent boundary layer interaction is investigated by simulating a supersonic flow over a 24° compression ramp. Incoming flow conditions are Mach number M = 2.9 and Reynolds number Re_θ = 2300 based on the inflow boundary layer momentum thickness. Synthetic turbulence is imposed at the inflow using a recycling and rescaling method. Supersonic outflow boundary conditions with non-reflecting treatment are used at the outlet and the top boundary. Non-slip condition is imposed at the wall, which is isothermal. The number of grid points used is 2432*x*256*x*160 in the streamwise, spanwise, and wall-normal directions, respectively. Fig. 3 illustrates an instantaneous field of the normalized temperature. Fig. 4 shows the mean wall-pressure distribution of the present DNS (STREAmS) together with the data from Wu and Martin [15] and Bookey et al. [16]. Again, results are in excellent agreement.

Conclusions

We developed an in-house solver for DNS of turbulent compressible flows over complex geometries in the framework of the generalized curvilinear coordinates. The code is based on high-order central finite-difference schemes hybridized with WENO reconstruction for shock capturing. The high-fidelity of the solver is ensured by the innovative stabilization technique of central schemes, which relies on physical principles related to the preservation of the total kinetic energy from convection, without the addition of any artificial viscosity. The optimization to run on multi-GPUs architectures makes our solver capable to perform large-scale DNS of a wide range of different flow configurations. The validation of the code was carried out by simulating a subsonic turbulent curved channel flow and a supersonic turbulent compression ramp flow. In both cases, the results proved to be in excellent agreement with previous numerical and experimental studies.

References

[1] Bernardini, M., Modesti, D., Salvadore, F., & Pirozzoli, S. (2021). STREAmS: a high-fidelity accelerated solver for direct numerical simulation of compressible turbulent flows. *Computer Physics Communications*, *263*, 107906. https://doi.org/10.1016/j.cpc.2021.107906
[2] Li, Y., Fu, L., & Adams, N. A. (2021). A low-dissipation shock-capturing framework with flexible nonlinear dissipation control. *Journal of Computational Physics*, *428*, 109960. https://doi.org/10.1016/j.jcp.2020.109960
[3] Piquet, A., Zebiri, B., Hadjadj, A., & Safdari Shadloo, M. (2020). A parallel high-order compressible flows solver with domain decomposition method in the generalized curvilinear coordinates system. *International Journal of Numerical Methods for Heat & Fluid Flow*, *30*(1), 2-38. https://doi.org/10.1108/HFF-01-2019-0048
[4] Pirozzoli, S. (2010). Generalized conservative approximations of split convective derivative operators. *Journal of Computational Physics*, *229*(19), 7180-7190. https://doi.org/10.1016/j.jcp.2010.06.006
[5] Kawai, S., & Lele, S. K. (2008). Localized artificial diffusivity scheme for discontinuity capturing on curvilinear meshes. *Journal of Computational Physics*, *227*(22), 9498-9526. https://doi.org/10.1016/j.jcp.2008.06.034
[6] Visbal, M. R., & Gaitonde, D. V. (2002). On the use of higher-order finite-difference schemes on curvilinear and deforming meshes. *Journal of Computational Physics*, *181*(1), 155-185. https://doi.org/10.1006/jcph.2002.7117
[7] Pirozzoli, S. (2011). Numerical methods for high-speed flows. *Annual review of fluid mechanics*, *43*, 163-194. https://doi.org/10.1146/annurev-fluid-122109-160718
[8] Chandravamsi, H., Chamarthi, A. S., Hoffmann, N., & Frankel, S. H. (2023). On the application of gradient based reconstruction for flow simulations on generalized curvilinear and dynamic mesh domains. *Computers & Fluids*, *258*, 105859. https://doi.org/10.1016/j.compfluid.2023.105859
[9] Gaitonde, D., & Visbal, M. (1999, January). Further development of a Navier-Stokes solution procedure based on higher-order formulas. In *37th Aerospace Sciences Meeting and Exhibit* (p. 557). https://doi.org/10.2514/6.1999-557
[10] Pirozzoli, S. (2011). Stabilized non-dissipative approximations of Euler equations in generalized curvilinear coordinates. *Journal of Computational Physics*, *230*(8), 2997-3014. https://doi.org/10.1016/j.jcp.2011.01.001
[11] Ducros, F., Ferrand, V., Nicoud, F., Weber, C., Darracq, D., Gacherieu, C., & Poinsot, T. (1999). Large-eddy simulation of the shock/turbulence interaction. *Journal of Computational Physics*, *152*(2), 517-549. https://doi.org/10.1006/jcph.1999.6238
[12] Spalart, P. R., Moser, R. D., & Rogers, M. M. (1991). Spectral methods for the Navier-Stokes equations with one infinite and two periodic directions. *Journal of Computational Physics*, *96*(2), 297-324. https://doi.org/10.1016/0021-9991(91)90238-G
[13] Moser, R. D., & Moin, P. (1987). The effects of curvature in wall-bounded turbulent flows. *Journal of Fluid Mechanics*, *175*, 479-510. https://doi.org/10.1017/S0022112087000491
[14] Brethouwer, G. (2022). Turbulent flow in curved channels. *Journal of Fluid Mechanics*, *931*, A21. https://doi.org/10.1017/jfm.2021.953
[15] Wu, M., & Martin, M. P. (2007). Direct numerical simulation of supersonic turbulent boundary layer over a compression ramp. *AIAA journal*, *45*(4), 879-889. https://doi.org/10.2514/1.27021
[16] Bookey, P., Wyckham, C., Smits, A., & Martin, P. (2005). New experimental data of STBLI at DNS/LES accessible Reynolds numbers. In *43rd AIAA Aerospace Sciences Meeting and Exhibit* (p. 309). https://doi.org/10.2514/6.2005-309