LES OF SECONDARY FLOW STRUCTURES DUE TO ENDWALL IN TURBINE FLOWS USING AN IN-HOUSE FLOW SOLVER

Sarp ER* AND Ayse G. GUNGOR†

Istanbul Technical University
Faculty of Aeronautics and Astronautics
Istanbul, TURKEY
*e-mail: ersa@itu.edu.tr †e-mail: ayse.gungor@itu.edu.tr

Key words: Large eddy simulation, Turbine flows, Secondary flow structures, Turbulence, Parallel CFD applications

Abstract. In-house large eddy simulation (LES) solver *lestr3d* is used for the investigation of the secondary flow structures formed in turbine flows due to the presence of endwall. Parallel performance of the solver shows very good speed-up performance with increasing processing units. Results of Taylor-Green vortex case is given to examine the effects of different schemes implemented to the solver. Preliminary results from the application of the solver to a turbine flow shows that the endwall related flow structures can be captured successfully. In the final paper, details of these secondary flow structures and their relation with aerodynamic losses will be discussed in more detail. Also the performance of the developed solver will be compared with an open source flow solver.

1 INTRODUCTION

Due to increasing demands in emission levels and efficiency, a more detailed approach is required for the design process of machinery. Turbines, at the heart of the energy production, becomes an important focus point. With the help of increasing computational power, new tools like LES and DNS becomes more and more applicable for the investigation of these type of flows[1, 2]. But considering the complex problem of secondary flow structures forming in the vicinity of the endwall, DNS is still too expensive to apply with the common computational resources. LES becomes prominent as it enables us to resolve the time dependent flow structures in turbine flows. Characteristics of turbulent structures forming inside turbines and their relation with energy losses can be analysed in detail by the increasing simulation capabilities [1, 3, 4].

In-house solvers became widely employed tools for aerospace applications both in academy and industry, leading to numerous solvers being developed. A modular, scalable, in-house LES solver is developed for the simulation of compressible, wall-bounded turbulent flows. In this study, our aim is to improve the abilities of the solver to better resolve the turbulent flow field and to investigate the secondary flow structures in turbine flows associated with endwall by using the developed solver. Losses related to these secondary flow structures will be discussed in the final paper. Besides that, the performance of the solver will be assessed.

2 SOLVER DETAILS AND PARALLEL PERFORMANCE OF *lestr3d*

The in-house solver *lestr3d* is written in FORTRAN language and solves compressible LES equations. Finite volume methodology is used to discretize the equations. The fluid is assumed to be Newtonian and an ideal gas. Compressible extension of the Smagorinsky model [5] and temperature gradient approach [6] is used for the calculation of the sub-grid scale(sgs) stress tensor and total enthalpy flux terms arising due to filtering process. Jameson-Schimidt-Turkel scheme is used for the calculation of the artificial dissipation in order to eliminate oscillations resulting from the central scheme employed in spatial discretization. Effect of the artificial dissipation on the resolution of the turbulent scales is analyzed in a previous study [8]. Time advancement is carried out with five-stage Runge-Kutta scheme. Boundary conditions are applied by using ghost cell methodology.

Two different numerical schemes for the calculation of convective fluxes are tested and they will be referred as M1 and M2 where averaging is used in M1 and Green-Gauss gradients are utilized [9] in M2. Taylor-Green vortex case is used for the comparison of these schemes. A cubical domain is used for Taylor-Green vortex case with the side length $2\pi L$ and periodic boundary conditions are applied in all directions. The initial velocity and pressure fields are given by the following equations;

$$u = V_0 sin(\frac{x}{L})cos(\frac{y}{L})cos(\frac{z}{L}), \quad v = -V_0 cos(\frac{x}{L})sin(\frac{y}{L})cos(\frac{z}{L}), \quad w = 0,$$

$$p = p_0 + \frac{\rho_0 V_0^2}{16}\left(cos(\frac{2x}{L}) + cos(\frac{2y}{L})\right)\left(cos(\frac{2z}{L}) + 2\right). \tag{1}$$

The Reynolds number of the flow is 1600, based on the reference velocity V_0, domain edge length L and kinematic viscosity of the fluid ν. And the Mach number is 0.11. Flow is initialized with vortices that break up into smaller structures and transition to turbulence occurs leading to a decay phase of isotropic turbulence. Simulation is run for 20 convective times(t_c) where the convective time is defined as $t_c = L/V_0$.

The total energy, E is tracked during the simulation. Apart from that, the dissipation rate associated with enstropy is calculated by using the relation, $D_{ens} = 2\nu\mathcal{E}$. Here the enstropy is calculated as $\mathcal{E} = \frac{1}{\rho_0\Omega}\int_\Omega \rho\frac{\omega\cdot\omega}{2}d\Omega$. This relation is exact for incompressible flows and additional terms due to compressibility can be neglected due to low Mach number of the present simulation. Figure 1(a) shows the change of the dissipation rate $(-dE/dt)$ with time. Results are compared with the study of DeBonis [10]. The agreement of the result obtained with M2 scheme is better compared to the M1 scheme when the same grid is used. Maximum dissipation occurs near $t = 9$ following the initial stage where small turbulent structures are formed as the large vortices initialized break-up with growing instabilities. M2 scheme captures the evolution of these small structures more successfully. M2 scheme is used in simulations presented in this study unless specified.

Speed-up $S = T_{ref}/T_N$, of the code can be seen from figure 1(b) tested up to 448 cores.

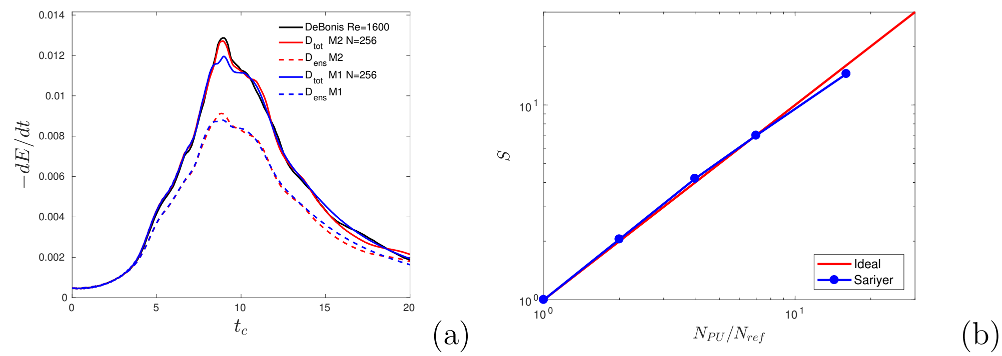

Figure 1: Change of kinetic energy dissipation (solid) and enstropy related dissipation D_{ens} (dashed) with convective time $t_c = L/V_0$ compared with results from DeBonis [10] (a) Speed-up of the code (b).

Speed-up with respect to the reference time T_{ref} is given, where T_{ref} is the time it takes to solve the test case with reference number of PUs $N_{ref} = 28$ which is the number of cores in a node supplied. It can be seen from the figure that the speed-up of the solver is excellent. Lowest speed-up performance is observed in test with 448 cores where 90% of the ideal speed-up is achieved.

3 APPLICATION OF *lestr3d* SOLVER TO SECONDARY FLOW STRUCTURES IN TURBINE FLOWS

In this section, the flow structures that are formed due to endwall will be investigated with *lestr3d* and their effects on the blade performance will be analysed. The T106 LPT blade geometry is considered for the examination of endwall effects. The Reynolds number of the flow is 80000, based on the inlet velocity U_0, chord length c and kinematic viscosity of the fluid ν. The flow enters the computational domain with an angle of 45° with respect to inlet plane normal vector. And the linear cascade geometry is attained by defining periodic boundary conditions in transversal direction. Thus the interaction of the secondary flow structures with neighbouring blades can be analysed. The Q criterion isosurfaces coloured by the spanwise velocity is shown in figure 2(a). The formation of horseshoe vortex in front of the blade can be seen. The pressure side leg of the vortex tends towards the suction side of the neighboring blade due to the pressure difference and passage vortex is formed as a result of this interaction. These secondary flow structures are not just transient structures but sustained during the normal operation conditions of the blade. Thus their relation with losses in the vicinity of endwall becomes important. Instantaneous results show that the secondary flow structures are being captured successfully with the developed solver. Figure 2(b) shows the distribution of the pressure coefficient on blade surface at $z/h = 0.5$ plane where h is the span of the blade. Preliminary statistics are in good agreement with the experimental and DNS data.

In the final paper, physics of these endwall related secondary flow structures and energy losses associated with the turbulent structures will be discussed in detail and the performance of *lestr3d* will be compared with an open source solver.

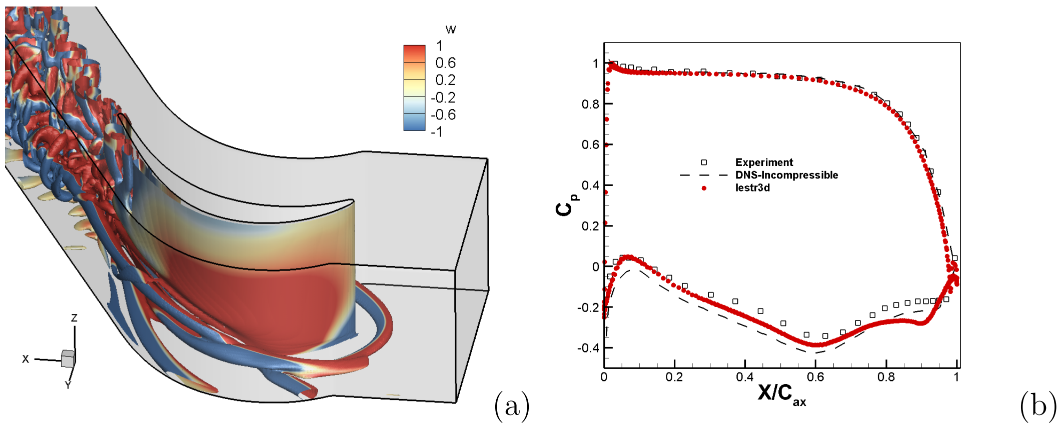

Figure 2: Q criterion isosurfaces coloured by the spanwise velocity fluctuations (a). Pressure coefficient distribution on blade surface at $z/h = 0.5$, compared with experiment and DNS results [11] (b).

REFERENCES

[1] Cui, J., Nagabhushana Rao, V. and Tucker, P., Numerical investigation of contrasting flow physics in different zones of a high-lift low-pressure turbine blade. *J. Turbomach.* 138(1), 011003 (2015).

[2] Michelassi, V., Chen, L.-W., Pichler, R., and Sandberg, R. D., Compressible Direct Numerical Simulation of Low-Pressure TurbinesPart II: Effect of Inflow Disturbances. *J. Turbomach.* 137(7), 071005 (2015).

[3] Michelassi, V., Wissink, J. G., Frhlich, J., and Rodi, W., Large-Eddy Simulation of Flow Around Low-Pressure Turbine Blade with Incoming Wakes. *AIAA Journal*, 41(11), 2143-2156 (2003).

[4] Raverdy, B., Mary, I., Sagaut, P., and Liamis, N., High-Resolution Large-Eddy Simulation of Flow Around Low-Pressure Turbine Blade. *AIAA Journal*, 41(3), 390-397 (2003).

[5] Erlebacher, G, Hussaini, M. Y., Speziale, C. G. and Zang, T. A. Towards the large-eddy simulation of compressible turbulent flows. *J. Fluid Mech., Vol 238*, p: 155-185, (1992).

[6] Menon, S. and Patel, N., Subgrid modeling for simulations of spray combustion in large-scale combustors, *AIAA Journal*, 44(4), 709-723, (2006).

[7] Karypis, G. and Kumar, V., A fast and high quality multilevel scheme for partitioning irregular graphs. *SIAM J. Scientific Comput.*, Vol 20, No 1, p: 359-392, (1999).

[8] Karahan, D.T., Er, S. and Gungor, A.G., Large eddy simulation of wall-bounded turbulent flows. *9th Ankara International Aerospace Conference(AIAC)*, Ankara, Turkey, AIAC-2017-047, (2017).

[9] Lowe, J., Probst, A., Knopp,T. and Kessler, R., Low-dissipation low-dispersion second-order scheme for unstructured finite-volume flow solvers. *AIAA Journal, Vol.* 54, No. 10., pp. 2961-2971 (2016).

[10] DeBonis, J., Solutions of the Taylor-Green Vortex Problem Using High-Resolution Explicit Finite Difference Methods. *51st AIAA Aerospace Sciences Meeting* January 710, Grapevine, Texas, U.S.A (2013).

[11] Wissink, J. G., Rodi, W., Direct numerical simulations of transitional flow in turbomachinery *J. Turbomach.* 110 668-678 (2004).