8th. World Congress on Computational Mechanics (WCCM8)
5th European Congress on Computational Methods in Applied Sciences and Engineeering (ECCOMAS 2008)
June 30 –July 5, 2008
Venice, Italy

ASSESSMENT OF LARGE-EDDY SIMULATION MODELS IN COMPLEX FLOWS USING UNSTRUCTURED MESHES

Oriol Lehmkuhl [1,2], Ricard Borrell [1,2], F.Xavier Trias [1] and C.D. Pérez Segarra [1]

[1] Centre Tecnològic de Transferència de Calor (CTTC)
Universitat Politècnica de Catalunya (UPC)
ETSEIAT, C. Colom 11, 08222 Terrassa (Barcelona), Spain
cttc@cttc.upc.edu, http://www.cttc.upc.edu

[2] Termofluids, S. L.
Magi Colet 8,
08204 Sabadell (Catalonia, Spain)

Key Words: *Turbulence modelling, Large-Eddy simulation models, unstructured mesh discretitzation schemmes, CFD parallel simulation*

ABSTRACT

Traditionally turbulence modelling of industrial flows in complex geometries have been solved using RANS models and unstructured meshes based solvers. The lack of precision of RANS models in these situations and the increase of computer power together with the emergence of new high-efficiency sparse parallel algorithms, make posible the use of more accurate turbulent models such as Large Eddy Simulation models (LES).

Recently, relevant improvements on turbulence modeling based on symmetry-preserving regularization models for the convective (non-linear) term have been developed. They basically alter the convective terms to reduce the production of small scales of motion by means of vortex-stretching preserving all inviscid invariants exactly. To do so, symmetries and conservation properties of the convective terms are exactly preserved. This requirement yields a novel class of regularizations that restrain the convective production of smaller and smaller scales of motion by means of vortex stretching in an unconditional stable manner, meaning that the velocity can not blow up in the energy-norm (in 2D also: enstrophy-norm). The numerical algorithm used to solve the governing equations must preserve the symmetry and conservation properties too. Since now, they have been successfully tested for a differentially heated cavity at high Rayleigh (Ra) numbers. At this stage, results using regularization models at relatively complex geometries and configurations are of extreme importance for further progress. The main idea behind this is to assess the validity of turbulence models in more realistic configurations, understand their limitations and finally improve them. Therefore, this is really a crucial issue since turbulence modeling ultimately becomes an essential tool for engineering applications.

In this paper a non-dissipative unstructured mesh scheme will be presented and different LES models will be tested. The selected models are: standard Smagorinsky model, dynamic Smagorinsky model, dynamic One-Equation model, and a symmetry-preserving Regularization model.

Four test cases will be simulated: two external flows and two internal flows. The first ones will be a square cilinder and a simplified Ahmed car, while the latter will be a plane impinging jet and a

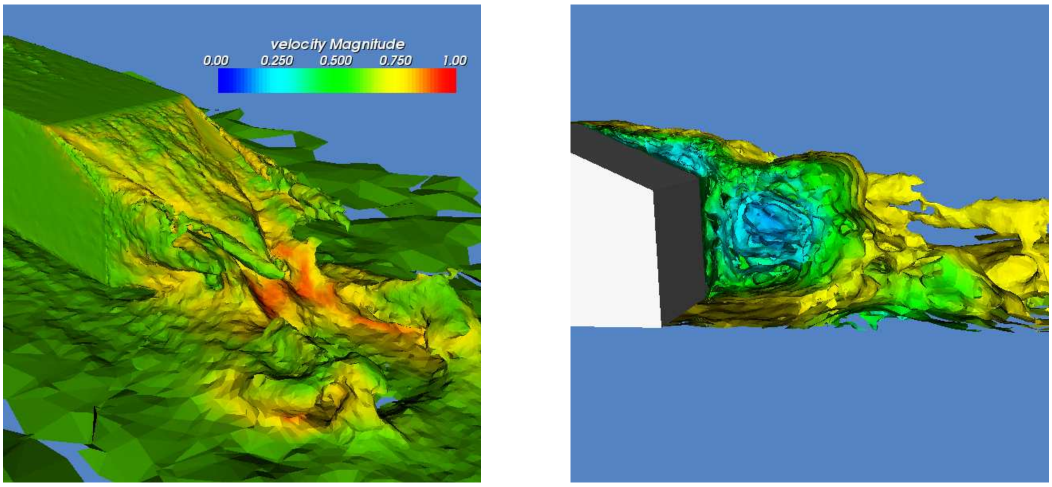

Figure 1: *External flow over a simplified Ahmed car with a Reynolds of* $7.68 \cdot 10^5$ *and a rear slant angle of* 25^o *with the dynamic Smagorinsky LES model Left: Instantaneous wake flow structure. Right: Instantaneous streamwise velocity fields in the symmetry plane (z=0.5m).*

differentially heated cavity of aspect ratio 4. These cases cover most of the flows that can be found in industrial situations. All the cases will be validated using experimental or DNS data.

The numerical experiments will be carried out by using the CFD code TermoFluids. TermoFluids is a new unstructured and parallel object-oriented CFD code for accurate and reliable solving of industrial flows. In all cases a explicit finite volume fractional-step based algorithm will be used. The pressure equation will be solved by means of a parallel Schur decomposition solver which is an efficient direct solver for loosely coupled PC clusters.

An illustrative result of the simulation of the external flow over a simplified Ahmed car with a Reynolds of $7.68 \cdot 10^5$ and a rear slant angle of 25^o is shown in the endosed figures. The simulation was carried out using the dynamic Smagorinsky LES model over a 3 millons control volume unstructured mesh on 18 CPUs.

REFERENCES

[1] F. X. Trias, M. Soria, A. Oliva, C. D. Pérez-Segarra. *Direct numerical simulations of two- and three-dimensional turbulent natural convection flows in a differentially heated cavity of aspect ratio 4*, J. Fluid Mech., Vol. 586, pp. 259–293, 2007.

[2] W. Rodi. *DNS and LES of some engineering flows*, Fluid Dynamics Research, Vol. 38, pp. 145–173, 2006.

[3] O. Lehmkuhl, C. D. Pérez Segarra, R. Borrell, M. Soria, A. Oliva. *TermoFluids: A new Parallel unstructured CFD code for the simulation of turbulent industrial problems on low cost PC Cluster*, Proceedings of the Parallel CFD Congress, Antalya (Turkey), May 2007.

[4] F. X. Trias, M. Soria, A. Oliva and R. Verstappen *Regularization models for the simulation of turbulence in a differentially heated cavity*, ECCOMAS CFD 06, Egmond aan Zee (The Netherlands), September 2006.

[5] H. Lienhart, S. Becker. *Flow and Turbulent Structure in the Wake of a Simplified Car Model*, SAE Paper no 2003-01-0656, 2003.