11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

TURBULENT THERMAL MIXING IN A T-JUNCTION: ADVANCED RANS VS. SCALE RESOLVING SIMULATION

Azzi Abbes[1], Nemdili Fadela[2] and Dellil Ahmed Zineddine[3]

[1] USTO-MB University, Oran, Algeria, BP 1505 El-Mnaouar, 31000, E-mail: azzi.abbes@yahoo.fr
[2] USTO-MB University, Oran, Algeria, BP 1505 El-Mnaouar, 31000, E-mail: turbo_fad@yahoo.fr
[3] Oran Senia University, Algeria, E-mail: sdellil@yahoo.fr

Key Words: *Turbulence models, SAS-SST, Two-Layer models, T-Junction.*

The configuration of interest here has been selected by the 15th ERCOFTAC Workshop on Refined Turbulence Modelling as a benchmark for turbulent convection mixing. The flow consists of air flow at 12°C entering a small duct of diameter B mixing with air flow at 60°C entering a larger duct of diameter 2B. The experimental tests were conducted by Prof. Masafumi Hirota from Department of Mechanical Engineering, Japan [1]. The configuration is similar to that used for film cooling of gas turbine blades but with a major difference relating to the width of the vertical branch, which is in film cooling smaller than that of the main channel and equal in the present application. One of the typical T-junction applications is found in the HVAC (Heating, Ventilating, and Air-Conditioning) unit used for automotive air-conditioning system.

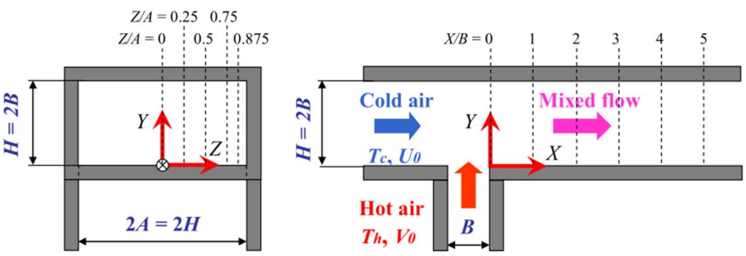

Figure1. Schematic diagram of the test channel

SOLUTION METHODOLOGY

The governing Navier Stokes equations are solved by the use of a three-dimensional finite-volume method that allows the use of arbitrary nonorthogonal grids, employing a cell-centered grid arrangement. A detailed description of the method is reported in Majumdar et al. [2]. Both RANS and scale resolving equations have been solved; in the later approach (inherently unsteady), use was made of Scale Adaptive Simulation model (SAS-SST).

RANS MODELLING DESCRIPTION

In the RANS context, the Reynolds-stress tensor is approximated within the context of the $k-\varepsilon$ turbulence model coupled with a one-equation model resolving the near-wall viscosity affected regions: two-layer approach. The two-layer DNS-Based $k-\varepsilon$ model (TLV) used here is a re-formulation of the so-called velocity-scale-based model (TLV) of Rodi et al. [4], in the sense that the turbulent kinetic energy is re-incorporated as a velocity scale. A detailed description of the model is reported in Azzi and Lakehal [5].

SCALE RESOLVING SIMULATION

SAS-SST turbulence model is built in frame of URANS strategy but it has the capability of resolving the turbulent spectrum in unsteady flow regions. According to Menter [6], the Rotta's k-kL turbulence model is well suited for a term-by-term modeling and shows interesting features compared to other approaches. Nevertheless, the weakest part made in this model is in neglecting the second velocity derivative and maintaining the third one. The model is then suited only for homogenous turbulence and need additional terms to be applied in the near wall zones. Menter [7] suggests then to replace problematic third velocity derivatives with the second one. The formulation proposed by Menter can operate in standard RANS mode, but has the capability of resolving the turbulent spectrum in unsteady flow regions. SAS models adapt the length scale automatically to the resolved scales of the flow field. The distinguishing factor in the model is the use of the von Karman length-scale, L_{vK}.

IMPLEMENTATION AND CASE SPECIFIC DETAILS

The computational domain is composed by two blocks representing the main channel and the branch. Only half of the physical domain is computed and a symmetry boundary is applied at z=0 plane applicable to RANS simulations only. In SAS-SST, the entire domain is considered. Conforming to the experimental test, the coordinate system is set with x downstream, y crossflow, and z spanwise, with the (x,y,z)=(0,0,0) origin downstream of the center of the vertical branch. Results will be provided along specified lines as showed in figure 1. Furthermore and according to computational evolution, results with SAS-SST computations will be presented.

REFERENCES

[1] Hirota, M. Mohri, E. Asano, H. and Goto, H. [2010], Experimental study on turbulent mixing process in cross-flow type T-junction, International Journal of Heat and Fluid Flow , 31, Issue 5, pp. 776-784.

[2] Majumdar, S. Rodi, W. and Zhu, J. T.B. [1992], Three-Dimensional Finite-Volume Method for Incompressible Flows With Complex Boundaries, ASME J. Fluids Eng., 114, pp. 496-503.

[4] Rodi, W. Mansour, N. N. and Michelassi, V. [1993], One-Equation Near Wall Turbulence Modelling with the Aid of Direct Simulation Data, ASME J. Fluids Eng., 115, pp. 196-205.

[5] Azzi, A. and Lakehal, D. [2002], Perspectives in Modeling Film Cooling of Turbine Blades by Transcending Conventional Two-Equation Turbulence Models, EASM J. of Turbomachinery, 124, pp. 472-484.

[6] Menter, F. R. and Egorov, Y., "The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 1: Theory and Model Description", Flow Turbulence Combust 85:113-138, 2010.

[7] Menter, F. R. and Egorov, Y., "The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 2: Application to Complex Flows", Flow Turbulence Combust 85:139-165, 2010.