

CFD INTERCOMPARISON EXERCISE WITHIN TRAJOS EUROPEAN RESEARCH NETWORK

**LOUKA P.^{1,2}, KETZEL M.³, SAHM P.², GUILLOTEAU E.⁴, MOUSSIOPOULOS N.²,
SINI J.-F.¹, MESTAYER P.G.¹, BERKOWICZ R.³**

¹ Laboratoire de Mécanique des Fluides CNRS - Ecole Centrale de Nantes, 1 Rue de la
Noë, P 92101, 44321 Nantes Cedex 3, FRANCE

² Laboratory of Heat Transfer and Environmental Engineering, Aristotle University of
Thessaloniki, PO Box 483, 54006 Thessaloniki, GREECE

³ Department of Atmospheric Environment, National Environmental Research Institute,
Roskilde, DENMARK

⁴ Institute of Hydromechanics, University of Karlsruhe, Karlsruhe, GERMANY

ABSTRACT

The results of a Computation Fluid Dynamics (CFD) intercomparison exercise that was performed by the working group on CFD modelling of the European research network of TRAJOS are presented. Several numerical models (CFX-TASCflow, CHENSI, CHENSI-2, MIMO, MISKAM) employing the widely used 'standard k-ε -model' were applied to well defined test cases and their results are compared. The cases treated were a two-dimensional single cavity, a single cube and a full-scale street canyon. Comparison of the model predictions with the available data is performed. The simpler study cases (single cavity and cube) demonstrate the level of agreement expected between similar codes and aims at improving and further developing the models. The discussion on the full-scale street case aims at explaining the differences along with giving some recommendations to CFD model users on how to calculate such complex flows and give useful suggestions to experimentalists on the concentration measuring position in order to achieve a more representative picture of air pollution dispersion in a street.