*Proceedings of the World Congress on Mechanical, Chemical, and Material Engineering (MCM 2015)*
*Barcelona, Spain, July 20 - 21, 2015*
*Paper No. 301*

# Numerical Study of Impinging Jet Flow with Application to Pressurized Themal Shock Situations.

**Michail Georgiou, Miltiadis V. Papalexandris**
Institute of Mechanics, Materials and Civil Engineering, Université catholique de Louvain
Place du Levant 2, B1348, Louvain-la-Neuve, Belgium
michail.georgiou@uclouvain.be; miltos@uclouvain.be

## Extended Abstract

In this talk, we present results from Large Eddy Simulations (LES) of an impinging cold turbulent jet to a hot, turbulent channel flow. This type of flow is encountered in various industrial processes and natural phenomena. Moreover, this flow is particularly relevant to the "Pressurized Thermal Shock" (PTS) phenomenon that may occur in pressurized water reactors of nuclear power-plants after a Loss-of-Coolant-Accident (LOCA). More specifically, due to a LOCA, the temperature in the cooling circuit faces a significant increase; this is counter-balanced by the injection of cold water. However, the sudden injection of a cold jet leads to transients in the temperature field that generate large thermal stresses. These streses can potentially affect the integrity of the circuit's pipes.

To numerically simulate the flows of interest, we consider the low-Mach number approximation of the compressible Navier-Stokes-Fourier equations without invoking the Boussinesq approximation. The numerical integration of the governing equations is performed with a two-stage, predictor-corrector time advancement algorithm that incorporates an Adams-Bashforth method for the convective and viscous fluxes. This algorithm also employs a projection method for the computation of the pressure field. According to it, the divergence of the momentum equation yields a Poisson equation for the pressure that is discretized and solved iteratively. Spatial discretization of differential operators in the governing equations is performed on a collocated grid. This is combined with a flux interpolation technique to avoid the pressure odd-even decoupling that is typically encountered when using such grids. Further, the viscous terms normal to the wall are solved implicitly, to increase the allowable time-step size.

For the subgrid-scale computation, the modified dynamic Germano model for varying density flows combined with the least square technique of Lilly is utilized. The unresolved scalar fluxes in the energy equation are described from a simple gradient assumption with a constant turbulent Prandtl number. In principle, the turbulent Prandtl number can also be calculated using the dynamic procedure. During our numerical experiments, however, it was predicted that this does not improve the quality of the simulations. In the initial set-up of our numerical study, the viscous sub-layer is fully resolved. However, the utilization of a wall function would dicrease substantialy the computational cost. Hence a study with respect to the potential use of a wall funtion is also performed within the framework of this study.

To accurately capture the properties of the turbulent field that emerges in the problem considered herein, appropriate boundary conditions (turbulent inflow-outflow) have to be chosen. In this study, the data for the two turbulent inflows are produced by two auxiliary simulations of turbulent channel flow. As regards the outflow boundary, we have opted for a convective boundary condition, which ensures mass conservation and suppresses spurious reflections off the numerical outflow boundary.

In our talk we present and discuss the statistics of the turbulent flow and heat transfer under consideration. Also, quantities regarding mixing rates and thermal stresses are presented and analysed. Our presentation concludes with a discussion on the effect of the angle of the incoming cold jet.