POISEUILLE-BENARD INSTABILITY IN RECTANGULAR DUCT WATER FLOW

C. Abid, C. Bonnefoi, M. Medale, F. Papini
Ecole Polytechnique Universitaire de Marseille, U.M.R CNRS N° 6595,
Technopole de Château-Gombert, 5 rue Enrico Fermi,
13453 Marseille 13. FRANCE

This paper deals with thermoconvective instabilities occurring in a horizontal rectangular duct water flow, heated from below (Poiseuille-Benard instability). Many experimental and theoretical results (restricted to moderate Reynolds and Rayleigh numbers for the linear stability analysis) have shown that the thermally stratified Poiseuille flow remains stable as long as the Rayleigh number (Ra) does not exceed a critical value Ra*. Beyond this value, the base flow becomes unstable and two kinds of thermoconvective structures, called “transversal rolls” and “longitudinal rolls”, may appear depending on the respective strength of the forced and induced flows. Mixed convection flows in a horizontal rectangular channel has nevertheless been widely studied for a while. During the first half of the 20th century, several researches on this subject attempted to explain certain meteorological phenomena. More recently, many related researches have been much more concerned with applications to technological processes like the cooling of electronic devices or Chemical Vapor Deposition in reactors. Therefore, most of these works are mainly focused on the heat transfer enhancement related to thermoconvective structures in the flow.

The aim of the present study is to investigate thermoconvective instabilities in Poiseuille-Bénard convection for a wide range of Re and Ra values, in order to better understand the basical mechanisms responsible for the instabilities to occur. This work has been undertaken with a combined approach linking experiments and numerical simulations.

Our experimental device (see figure 1) is a horizontal rectangular duct uniformly heated from below. This channel is 1.4 cm high, 2.6 cm wide and 2 m long; its 3 mm thick walls are made of Plexiglas. The first part of the channel is designed to enable the isothermal inlet fluid flow to hydraulically establish itself so that a Poiseuille type flow is obtained at the outlet of this zone. Next to that part is the central testing zone, which is heated from below on 0.5 m, through the lower wall made of copper. This bottom wall also hosts an electrical resistance which delivers the prescribed heating when a direct electric current is applied between the input and output terminals (Joules effect heating). The applied heat flux is uniform along the cross section throughout the testing zone and can be tuned from 76 $W.m^{-2}$ to 30 $kW.m^{-2}$. Finally the experimental device enables us to measure local fluid temperature (by means of K-type thermocouples) in several longitudinal locations and various heights in the duct. Moreover the fluid flow structures and the velocity field are obtained using the PIV. Several series of measurements have been carried out in order to cover a large range of fluid flow rates (Reynolds number) and applied heat fluxes (Rayleigh number).

On the other hand, a numerical study has been undertaken. The numerical model consits in the solution of the incompressible Navier-Stokes and energy equations. Nevertheless, since we assume in a first stage the Boussinesq approximation stands, it implies to only consider moderate heating configurations in the computations. The 3D numerical model we have developed to solve the coupled fluid flow and heat transfer problem is based on the finite element method. First of all, a segregated approach is used to build up the integral forms associated on one hand with the incompressible fluid flow problem, and on the other hand with the heat transfer problem. The former is written in a primary variable formulation and is solved using an unconditionally stable projection algorithm. Then, the spatial discretisation is achieved using tri-quadratic hexahedral finite elements for the velocity and temperature variables, whereas a piecewise tri-linear approximation is used for the pressure. At each time step, the three algebraic systems related to the momentum, incompressible projection and energy conservation, are solved with iterative solvers provided in the Pestc toolkit, running high performance massively parallel computers. The two basic configurations we have considered in our preliminary numerical simulations mainly differ in the way to take into account the heating system along the lower horizontal wall: constant temperature or constant heat flux. For both cases we have carried out fully transient numerical computations. In the isothermal heating model, we have found a threshold, which distinguishes the configurations where a steady state solution exists, from those where only unsteady solution was found (periodic behaviour close above the threshold, and dynamically more complex when going further). On the other hand for the constant heat flux model, we have also found a transition in the dynamical regime, but unlike in the former case we haven't found any periodic behavior yet.

The obtained experimental results allow us to highlight various regimes and in particular a domain where the thermoconvective instabilities occur; furthermore a stability diagram has been established (see figure 2). On the other hand, the computations have helped us to identify and better understand the possible basic mechanisms leading to the quite complex observed fluid flow, which results in the combination of the forced fluid flow and the buoyancy induced one. In the range of parameters considered in the computations (Re=200, moderate Ra numbers to satisfy the Boussinesq approximation), the fluid flow structure is mainly composed of two longitudinal rolls, which interact dynamically each other when the steady-unsteady threshold is overcome.

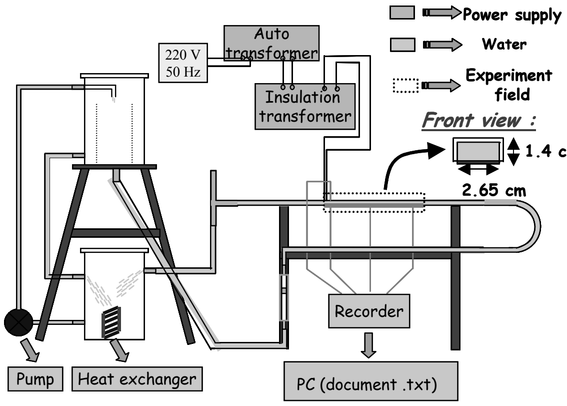


Figure 1: Ex perimental setup

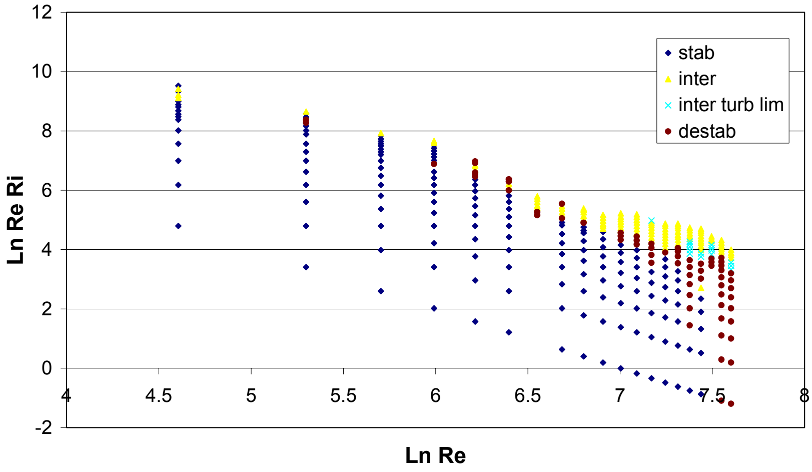
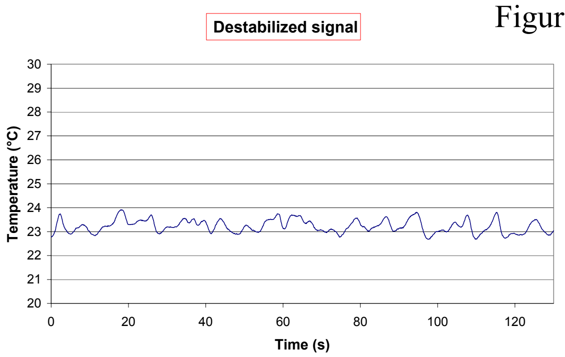
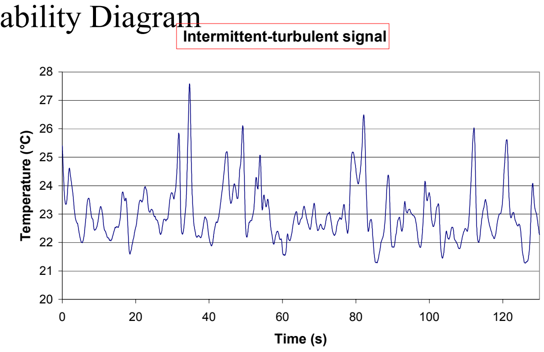
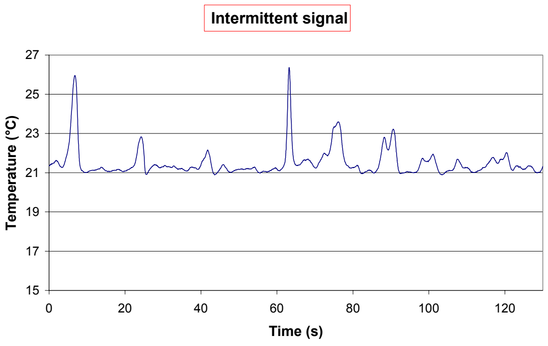
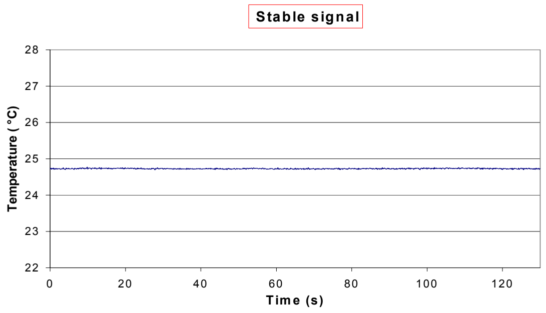


Figure 2 : Stability Diagram


Destabilized signal


Intermittent-turbulent signal


Intermittent signal


Stable signal