EXPERIMENTAL AND NUMERICAL INVESTIGATION OF TAYLOR FLOW

Delphine Laboureur, Belgium
Supervisors: J. Anthoine & B. Tóth

INTRODUCTION & OBJECTIVES:

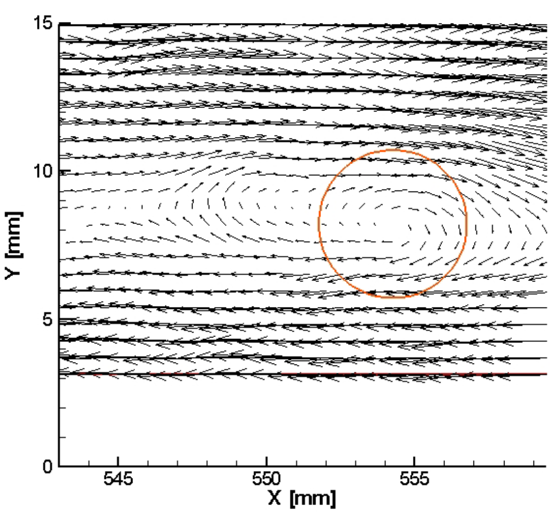
The flow observed in the boosters of the Ariane 5 launcher can be represented theoretically by the Taylor flow. Different flow regimes exist inside the boosters, depending on the combustion time: laminar, transitional and turbulent. This project will focus on flow instabilities coming from the Taylor flow using PIV technique. For that purpose, a setup consisting of a rectangular shape channel, where air is introduced by the bottom wall thanks to a porous plate, has been made and validated last year within a TFE project (D. Laboureur). The objectives of the present project are triple. First, adapt the PIV technique to use it in the setup and obtain a full view of the channel. Then, investigate the different regimes of the Taylor flow which depend on the injection velocity and the channel height and finally make numerical simulation to support the previous investigation by following chronologically the instantaneous phenomenon.

METHODOLOGY:

Concerning the use of PIV technique, a special seeding generator was built and used to inject the particles through a tube inside the stagnation chamber. The reflection of the laser sheet from the porous plate was avoided by using a laser sheet of constant height coming from the end side of the channel. The measurement campaign using PIV technique is based on the thesis of Griffond in order to choose the parameters that reproduce the 3 different flow regimes. Four configurations were tested, 2 heights of the channel (23mm and 33mm) and two injection velocities (1 and 1.5m/s). For each of them, 5 fields of view were measured.
Concerning the numerical simulations, 3 cases were computed with 3 different channel heights (13, 23 and 33mm) and injection velocities (respectively: 1, 1.5 and 2m/s). The commercial software Fluent was used, performing LES simulations on a 3D domain with periodic condition in the third direction to simulate a 2D behavior. The mesh is following the dimensions of the experimental setup in length and height. The inlet conditions were based on hot wire measurements performed close to the porous plate.

RESULTS:

Either from the CFD or PIV data the mean flow was computed and compared with the Taylor flow theory, showing a good agreement. The statistical analysis of the detected vortices (like the ones of figure 1) using a wavelet tool and the turbulence intensity profiles showed that the flow behavior is changing with the channel height, being laminar for 33mm, transitional for 23mm and turbulent for 13mm. It has been observed from the CFD vorticity fields that the structures are emerging from the injection plane and rolling up to form a vortical structure that will dissipate into a cluster of smaller vortices when traveling to the end of the channel. The injection velocity seems to have an influence on the development of structures since it is more difficult to see an effect on the flow regime. The comparison of the CFD results with the PIV allowed validating them.



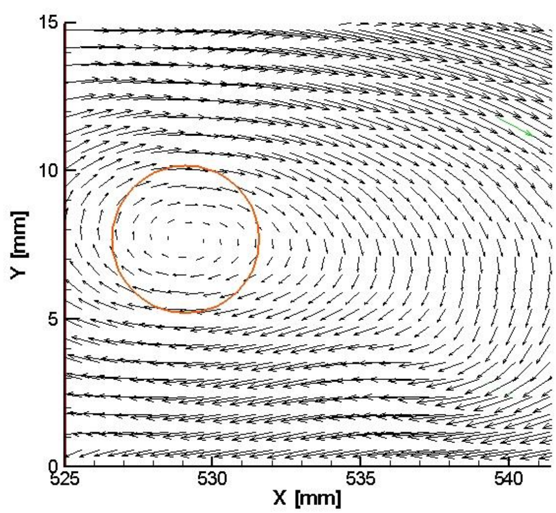


Figure 1: Detected vortices in a PIV (left) and CFD (right) instantaneous field