EXPERIMENTAL AND NUMERICAL INVESTIGATION OF THE FLOW FIELD INSIDE A SOLID PROPELLANT BOOSTER IN PRESENCE OF SLAG ACCUMULATION

Erinc Erdem, Turkey
Supervisors: J. Anthoine, B. Tóth, M. Lema

Solid propellant motors with submerged nozzles have to face, during combustion, the formation of a cavity around the internal part of the nozzle lips. In addition the inhibitors are present in the internal geometry in order to prevent combustion between the blocks of propellant acting as thermal protectors and vortices are shed when flow passes over the inhibitors causing the phenomenon so called Obstacle Vortex Shedding especially prior to the end of combustion. Droplets of liquid alumina, a product of the combustion, are interacting on one hand with the vortices generated by the inhibitors, and on the other hand with the nozzle, creating a liquid puddle in the cavity, called slag.

The aim of this project was to assess both experimentally and numerically this phenomenon, strictly speaking what is going on inside a simplified 2-Dimensional test section, which has been developed at VKI. This geometry includes an inhibitor and a cavity with a nozzle. The experimental investigation has taken place using a non-intrusive optical velocity measurement technique called Particle Image Velocimentry (PIV) in L11 vertical wind tunnel. On the other hand the numerical investigation has been carried out using a commercial Computational Fluid Dynamics (CFD) package for this test geometry.

PIV measurements are realized in the new facility for one velocity and one geometrical configuration (one inhibitor position, one cavity volume). There is no liquid droplet involved but the accumulated liquid slag is modelled as solid wall in the cavity. A significant number of PIV data has been acquired for this single test case to generate a detailed database for validation of the numerical simulation in the whole section.

The numerical simulation is performed using the commercial CFD software called CFD-RC with the help of some simplifications such as single-phase flow with no liquid interaction in 2D. Few turbulence models are used to simulate the flow phenomenon inside this geometry and they have shown good agreements with PIV experiments in the stream-wise direction excluding the cavity. In the cavity all the models underestimate the motion in stream-wise direction. The laminar case is also used for preliminary analysis and checking the computational grids. A quantitative comparison of all turbulent cases with experimental data at locations x=4h and 8h is shown below.

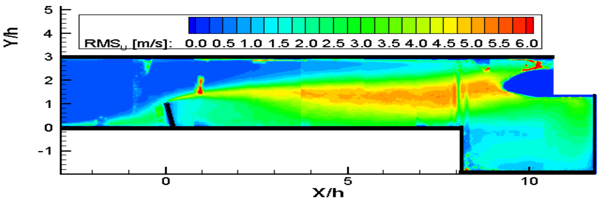

Figure 1 the domain of investigation in test section and Rms contours in stream-wise direction from PIV

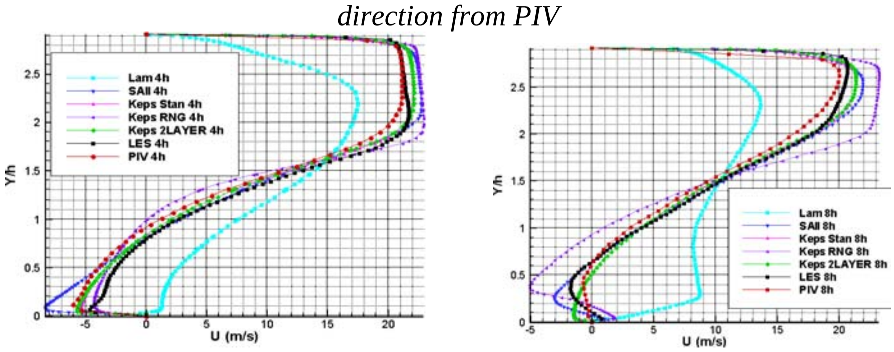

Figure 2 the stream-wise (u) velocity comparison at x=4h and x=10h respectively