EXPERIMENTAL MAPPING OF VELOCITY FLOW FIELD IN CASE OF L.O.V.A INSIDE STARDUST FACILITY

C.Bellecci[1], P.Gaudio[1], I.Lupelli[1], A.Malizia[1], M.T.Porfiri[2],R.Quaranta[1], M.Richetta[1]

[1]*EURATOM-Faculty of Engineering, University of Rome "Tor Vergata"*
Via del Politecnico 1, 00133 Rome, Italy;
[2]*ENEA Nuclear Fusion Tecnologies, Via Enrico Fermi , 45 I-00044 Frascati, Italy*

Introduction

Fusion power is a promising long term candidate to supply the energy needs of humanity. An International Thermonuclear Experimental Reactor (ITER) to generate 1500 MW has been designed, but the high cost of construction led to work on a reduced scale option. Plasma physics effects and (Plasma Materials Interactions) PMIs, that are only partially observed or accessible in present day experiments, will become important. Higher heat loads, more intense transient heating events [i.e. edge localized modes (ELMs), disruptions and Vertical Displacements Events (VDEs)], and the predicted magnitude of plasma facing component (PFC) damages, by melting and evaporation, are critical issues. A recognized safety issue for future fusion reactors (like ITER) fueled with deuterium and tritium is the generation of sizeable quantities of dust. The loss of coolant accidents (LOCA), loss of coolant flow accidents (LOFA) and loss of vacuum accidents (LOVA) are types of accidents that may jeopardize the components and the plasma vessel integrity and cause dust mobilization, risky for workers and public. In order to analyze dust resuspension in case of LOVA a small facility is been developed, STARDUST (Small Tank for the Aerosol Removal and DUST), in the ENEA laboratories of Frascati (by Fusion Technology Department) that represents a scaled section the ITER Vacuum Vessel. For the reasons described above, several experiments have been conducted with STARDUST facility in order to reproduce a low pressurization rate (300 Pa/s) LOVA event in ITER due to a small air leakage, for two different positions of the leak, at the equatorial port level (Valve A) and at the divertor port level (Valve B), in order to perform flow field velocity experiments. These experiments are necessary to evaluate the velocity flow field behavior, in case of a LOVA simulation, in several points inside STARDUST, because velocity amplitude in case of a LOVA is strictly connected with dust mobilization phenomena. Afterwards FLUENT was used to simulate the flow behavior for the same LOVA scenarios used during the experimental tests.

Experimental campaign and results

To measure the punctual velocity flow values a XCQ-093-2PSI-D Kulite pressure transducer has been used. The pressure transducer works in a differential mode.

To measure the punctual flow field velocity the Bernoulli's equation cannot be applied because, at higher speeds, the compressibility of air has to be taken into account. In this case, a more complex equation must be used:

$$v = \sqrt{\frac{2\gamma R\overline{T}}{M(\gamma-1)}\left[\left(\frac{P_\Delta + P_s}{P_s}\right)^{\frac{\gamma-1}{\gamma}} - 1\right]}$$

Where:

- **γ** : ratio of the fluid specific heat at constant pressure to the fluid specific heat at constant volume (c_p/c_v) and is approximately *1.4* for air;
- **R** : universal gas constant (8,314 J K^{-1} mol^{-1}) ;
- $\overline{T}$: mean temperature ;
- **M :** air molecular mass (28,968 g/mol);
- **P_s** : static pressure ;
- **P_Δ**: differential pressure ($P_\Delta = P_T - P_s$).
- **P_T** : total pressure

The differential pressure is automatically measured with the pressure transducer.

The punctual velocity values have been measured at several distances from inlet valves A and B:

- x=0
- x=6 cm;
- x=15 cm;
- x=45,5 cm
- x=80,5 cm.

For the distances x= 6 cm and x=15 cm several measurements on z-y plane have been conducted.

All the experiments have been carried out at room temperature condition (T_{wall} of 25° C) with inlet from both valves (A and B). Each set of experiments has been repeated 3 times and the mean values acquired have been analyzed.

The most important considerations are the following ones:

- The difference of velocity trend measured in the points on z-y plane are in the order of 10%, so these experiments results are negligible if compared to those done in the x-y plane (z=0) and will not be taken in account for final comparison and consideration;
- <u>Valve A</u>: there is a significant decrease of velocity magnitude in proximity of H =24 in particular it has been observed at distances of 6, 15, 41.5 centimeters from valve A;
- <u>Valve B</u>: The most important consideration is that there is a significant decrease of velocity magnitude in proximity of H =1.5, in particular it has been observed at distance of 6, 15, 41.5 centimeters from valve B;
- The variation obtained by moving the pressure transducer from a distance of 15 cm from valves to other distances are of the order of ±5% (with the exception of the two positions in front of the valves) and in good approximation we can avoid:
    - to detect velocity trend at that distance;

and

- to consider these values for final considerations.

Numerical simulations

All the simulation of the thermal and flow field were performed using the commercial software FLUENT based on the finite volume method. The governing equations in a compressible turbulent flow can be written as follows in which gravity effects are included. The implemented fluid-dynamic model is based on the fully compressible formulation of the continuity equation and momentum equations:

$$\frac{\partial p}{\partial t}+\nabla\left(\rho v\right)=S_m$$

$$\frac{\partial}{\partial t}\left(\rho\vec{v}\right)+\nabla\cdot\left(\rho\vec{v}\vec{v}\right)=-\nabla p+\nabla\cdot\mu\left[\left(\nabla\vec{v}+\left(\nabla\vec{v}\right)^{T}\right)-\frac{2}{3}\nabla\cdot\vec{v}I\right]+\rho\vec{g}+\vec{F}$$

Turbulence is a main key for this kind of problems. The aim of this work is to get a preliminary overview of the fluid dynamics behavior of a LOVA event. The RNG-based $k-\varepsilon$ model has been chosen because its robustness and wide spread use. Also, the model chosen improves the accuracy for rapidly strained flows than the standard $k-\varepsilon$ model.

The velocity magnitude near the inlet section at the beginning of a LOVA event is relevant for the dust mobilization, about 215m/s. Fig. 1 illustrates the velocity magnitude in the whole domain (1s) for the inlet A and B scenarios.

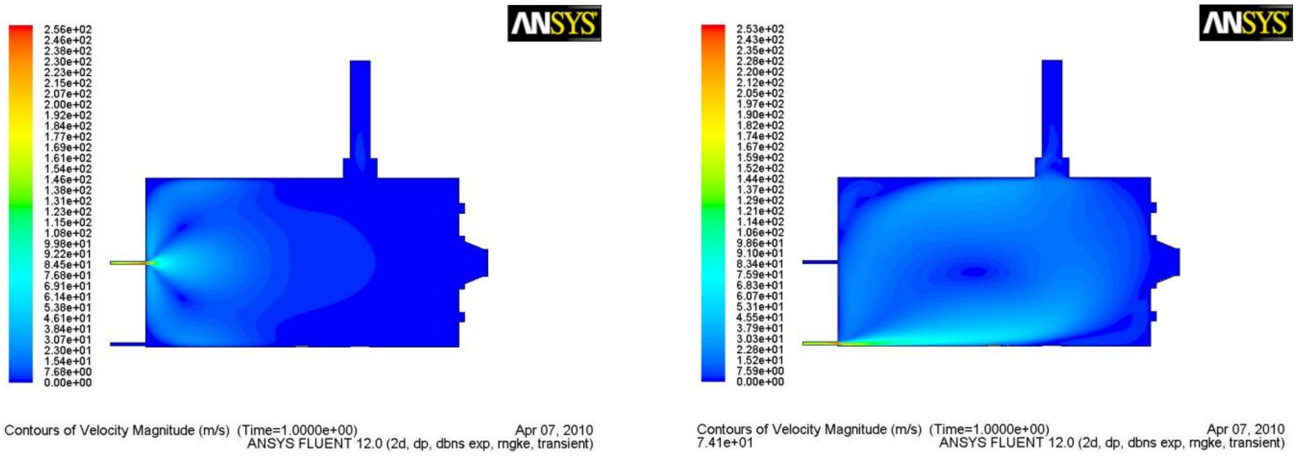


Fig. 1 : *Velocity magnitude (1s)*

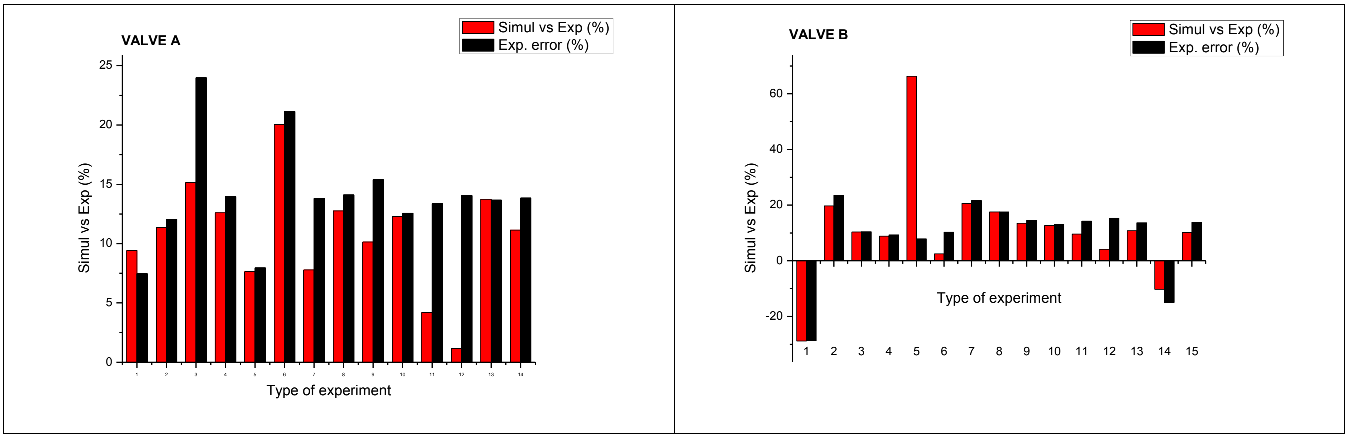
It can be observed that there is a recirculation zone for the valve B inlet.

Conclusion

It can be observed that exist an overestimation of the expected velocity magnitude for the valve A at about 2-3 s. This effect could be a consequence of the direct air fast transient impact that causes an excessive P_Δ on pressure transducer. This excessive P_Δ reduces, for the successive time steps, the measurement capability (probably saturation effect).

An underestimation of the expected velocity magnitude for the valve B can be observed as a consequence of finite dimension of the transducer respect to the infinitesimal dimension of the simulation virtual probe. The recirculation zone for the valve B can reduce the P_s magnitude acquired by the pressure reference tube and consequently increases P_Δ which is directly proportional to the velocity magnitude.

For all the experiments analyzed the percentage variation between the maximum numerical velocity values and the experimental ones has been calculated. These percentages have been then compared with the experimental error values (Fig.2) and in all the cases the simulation results show that the velocity magnitude at different points seems to be in good agreement with the experimental results.



A numerical model is able to describe a LOVA event is an important tool to support a fusion device. The same model has been used to predict the velocity field at the aim to compare for different experimental configuration during a LOVA event in the whole domain. Therefore, experimental activities and numerical simulation campaigns have been carried out in strong correlation in order to both understand the capabilities of computational codes and predict correctly the characteristics of the flows during a LOVA event.

- It can be observed that there is a recirculation zone for the valve B inlet. The flow pattern is more complex in valve B inlet case whereas in the case of valve A inlet it is less complex because of the geometry constraints.
- During this study, it was shown that the coupling of real gas assumption and RNG-based k-ε turbulence model could be used to simulate the LOVA event better than the real gas based Standard k-ε and the ideal gas based k-ε. Based on these studies, it can be seen that there is a lot of potential for the FLUENT CFD software package for developing a more complex 3D model of STARDUST or implementing dust particles tracking in the model.
- The simulation results show that the velocity measurements at different points are in seems to be in good agreement with the experimental results.