CHARACTERIZATION OF DIVERTOR INFLUENCE IN CASE OF LOVA: CFD ANALYSIS OF STARDUST EXPERIMENTAL FACILITY

C. Bellecci¹, P. Gaudio¹, I. Lupelli¹, A. Malizia¹, M. T. Porfiri², R. Quaranta¹, M. Richetta¹

¹EURATOM-Faculty of Engineering, University of Rome “Tor Vergata”; Via del Politecnico 1, 00133 Rome, Italy
²ENEA Nuclear Fusion Technologies, Via Enrico Fermi, 45 I-00044 Frascati, Italy

RESEARCH TARGETS

Intense thermal loads during the normal operation and anomalous events like Edge Localized Modes (ELMs), Plasma Disruptions (PD) and Vertical Displacement Events (VDEs) are expected to produce substantial quantities of particulate material in the form of dust, flakes, and spalled deposits from erosion of plasma facing components. This dust, mainly accumulated close to the divertor zone, can be mobilized outside the Vacuum Vessel (VV) in case of loss of vacuum accident (LOVA) threaten public safety because it may contain tritium, may be radioactive from activation products, and may be chemically reactive and/or toxic. The aim of this work is to analyze the velocity values in several points inside the facility STARDUST (Small Tank for Aerosol Removal and Dust) and compare results with numerical model in order to investigate and describe the potential impact on ITER safety analyses.

STARDUST DESCRIPTION

STARDUST is a stainless steel horizontal cylinder (with an internal volume of 0.17 m³) closed by two lids [1]. STARDUST is equipped with an automatic data acquisition system that allows the control of internal pressure, wall temperature and air flow inlet in order to carry out the experiments at the desired initial conditions (initial pressure 100 Pa and lateral walls temperature 25 °C). When the initial conditions are reached, the control system opens the flow meter inlet valve and then the air flows inside the tank with a flow rate of 27 l/min necessary to achieve a pressurization rate of 300 Pa/s. Air inlet simulates the loss of vacuum event. The air flow passes through two valves positioned in the back lid respectively at the middle of the tank, where the vacuum vessel ports are located, and at the bottom of the tank, where the divertor is located in a fusion plant [1,2,3,4,5]. The experiments have been carried out introducing a semi cylindrical obstacle made of stainless steel, which simulates the presence of structures, in particular the divertor, inside the ITER VV [4,5].

EXPERIMENTAL CAMPAIGN

In the experimental campaign has been evaluated punctual flow velocity values obtained in two different internal conditions:

- Experiments without obstacle (Vx_WO where X maybe A or B inlet valve);
- Experiments with obstacle, in order to understand better the influence of structures (like divertor) present inside STARDUST facility;

The experiments with obstacle has been conducted by measuring point velocity air values in three different positions:

- Under obstacle (UO)
- Inside obstacle, under the bridge (IOUB)
- Inside obstacle over the bridge (IOOB)

In order to keep in account the influence of the space between limiter and divertor, as in the VV of ITER, a slit has been realized on the obstacle (at bridge level) and experiments with pressure transducer inside the obstacle have been repeated. The velocity values have been measured by mean of differential pressure transducer located inside STARDUST in the positions that have been explained before [6]. The pressure transducer measures the difference of pressure (PA) between a pressure reference tube (static pressure Ps) and the pressure measured by the head of sensor (total pressure Pt). When the initial conditions inside the tank (internal pressure of tank 100 Pa, wall temperature of 20-25 °C) have been reached the PC
controller allows the acquisition (with a frequency of 50 Hz). For the calculation of velocity magnitude the following equation has been used:

\[ v = \sqrt{\frac{P}{\rho}} \]  

Where:
- \( \gamma \) : ratio of the specific heat of the fluid at constant pressure to the specific heat of the fluid at constant volume \((c_p/c_v)\) and is approximately 1.4 for air;
- \( R \) : universal gas constant
- \( T \) : mean temperature measured by internal thermocouples;
- \( M \) : air molecular mass \((28.968 \text{ g/mol})\);

**GEOMETRY AND BOUNDARY CONDITIONS OF NUMERICAL MODEL**

The model includes the primary geometrical features of STARDUST. The initial conditions of the tank and the air inlet (mass flow inlet) are showed in the following figure 1:

<table>
<thead>
<tr>
<th>Inlet Fluid Properties and Vessel Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Molar mass ((\text{kg/mol}))</td>
</tr>
</tbody>
</table>

**Inlet conditions**

- Inlet temperature \(T_i\) : \(25 \text{ °C} \pm 0.1 \text{ °C}\)
- Inlet pressure \(P_i\) : 2.8 bar \(= 280,000 \text{ Pa}\)
- Inlet density \(\rho_i\) : 1.272 g/m³
- Inlet specific heat capacity at constant pressure \((c_p)\) : \(564.2 \text{ J/kgK}\)
- Inlet specific heat capacity at constant volume \((c_v)\) : \(77.2 \text{ J/kgK}\)
- Inlet \(c_p/c_v\) ratio \(\gamma\) : 1.424
- Inlet dynamic viscosity (Sutherland) \(\eta_0\) : \(1,338 \times 10^{-5} \text{ kg/m s}\)
- Inlet speed of sound \(a_0\) : 343.1 m/s
- Inlet critical pressure \(P_c\) : 0.06804 bar

**Vessel conditions**

- Vessel air temperature \(T_v\) : \(37 \text{ °C} \pm 0.1 \text{ °C}\)
- Vessel air static pressure \(P_v\) : 0.3865 bar \(= 38,650 \text{ Pa}\)
- Vessel air density \(\rho_v\) : 0.00014 g/m³
- Inlet specific heat capacity at constant pressure \((c_p)\) : \(970.8 \text{ J/kgK}\)
- Inlet specific heat capacity at constant volume \((c_v)\) : \(675.6 \text{ J/kgK}\)
- Vessel air \(c_p/c_v\) ratio \(\gamma_v\) : 1.425
- Vessel air dynamic viscosity (Sutherland) \(\eta_1\) : \(1,843 \times 10^{-5} \text{ kg/m s}\)
- Vessel air speed of sound \(a_1\) : \(340.4 \text{ m/s}\)
- Vessel air critical pressure \(P_c\) : \(1,001.3 \text{ Pa}\)
- Vessel wall temperature \(T_w\) : \(207 \text{ °C} \pm 0.1 \text{ °C}\)

**TURBULENCE MODELING**

The model also takes in account the transient effect due to the delays introduced by the control apparatus and the valves (figure 2):
We have utilized for turbulence modeling the RNG (Renormalization Group Theory) k-ε model. The RNG k-ε model was derived using a rigorous statistical technique (called renormalization group theory). It is similar in form to the standard k-ε model but provides an analytically-derived differential formula for effective viscosity that accounts for low-Reynolds-number effects. The turbulent viscosity is calculated as:

\[
d\left( \frac{2}{3} \right) \frac{\nabla k}{\nabla \cdot k} - C_w \frac{\mu}{\nu} = 1.72 \left( \frac{\rho}{\mu} \right)^{1.2}
\]

where \( \mu \) is the molecular viscosity, \( C_v \) is a constant, and \( \frac{\nabla \cdot \omega}{\mu} \) is the ratio of turbulent to molecular viscosity:

\[
\frac{\nabla \cdot \omega}{\mu} = C_v, \quad \frac{\mu}{\nu} = \frac{1}{\rho}
\]

here \( \rho \) is the density and \( C_\mu \) is an empirical constant. Effective use of this feature does, however, depend on an appropriate treatment of the near-wall region. We have used a two-layer model for enhanced wall treatment to specify both \( C_v \) and the turbulent viscosity in the near-wall cells. A convection heat transfer coefficient is applied to all chamber walls, derived from the Nusselt number (Nu) correlation [7]. The entire domain, considered the inlet conditions, is initialized to a low value for turbulent kinetic energy, and a high value for the dissipation rate, in order to avoid the problem of high turbulent viscosities at the beginning of the simulation (see figure 1).

RESULTS

Initial modeling efforts have focused on short fill times (~ 3s). A section of the tank characterizing the flow is shown in figure 3.

Experimental and model results are compared in figure 4.
CONCLUSIONS AND FUTURE WORK
The model matches closely the experimental data with a slightly faster fill time. Simulations and experiments showed that the presence of an obstacle in STARDUST influence the dust’s mobilization solely in case of leakage at divertor ports level (valve B). This result matches with experiments done in 2007 experimental campaign [4,5] in which we have measured the quantity of dust resuspended in case of LOVA. The percentage of dust resuspended, that simulates a LOVA at divertor level is lower in case of obstacle inside the chamber [4,5]. These results matches with experimental and numerical results obtained in the current work. In fact the velocity values measured without obstacle (VB_WO) are higher than velocity values measured with obstacle (VB_WO). The 3D simulation of STARDUST filling matches with experimental results for fast transients (~ 1.5 s). Because of the limitation of time simulation the model does not predicts the oscillating flow after the vessel fills. Future work will focus on obtaining accurate results for longer transients, resolving flow features such as shocks and boundary layers, and implementation of dust resuspension models to accompany the Lagrangian discrete phase transport models. These discrete phase models include most of the transport phenomena relevant for ITER, including drag, gravitational, buoyant, and thermophoretic forces.

REFERENCES