



ELSEVIER

Contents lists available at ScienceDirect

Fusion Engineering and Design

journal homepage: www.elsevier.com/locate/fusengdes

Large eddy simulation of Loss of Vacuum Accident in STARDUST facility

Miriam Benedetti^a, Pasquale Gaudio^a, Ivan Lupelli^a, Andrea Malizia^{a,*},
 Maria Teresa Porfiri^b, Maria Richetta^a

^a Associazione ENEA EURATOM Quantum Electronics and Plasma Physics Research Group, Department of Industrial Engineering, University of Rome "Tor Vergata," Via del Politecnico 1, I-00133 Rome, Italy

^b Associazione ENEA EURATOM Nuclear Fusion Technologies, Via Enrico Fermi 45, I-00044 Frascati, Rome, Italy

HIGHLIGHTS

- Fusion safety, plasma material interaction.
- Numerical and experimental data comparison to analyze the consequences of Loss of Vacuum Accident that can provoke dust mobilization inside the Vacuum Vessel of the Nuclear Fusion Reactor ITER-like.

ARTICLE INFO

Article history:

Available online 20 February 2013

Keywords:

Nuclear fusion safety
 Loss of Vacuum Accident
 Dust mobilization
 Experiments
 Numerical simulation
 LES

ABSTRACT

The development of computational fluid dynamic (CFD) models of air ingress into the vacuum vessel (VV) represents an important issue concerning the safety analysis of nuclear fusion devices, in particular in the field of dust mobilization. The present work deals with the large eddy simulations (LES) of fluid dynamic fields during a vessel filling at near vacuum conditions to support the safety study of Loss of Vacuum Accidents (LOVA) events triggered by air income. The model's results are compared to the experimental data provided by STARDUST facility at different pressurization rates (100 Pa/s, 300 Pa/s and 500 Pa/s). Simulation's results compare favorably with experimental data, demonstrating the possibility of implementing LES in large vacuum systems as tokamaks.

© 2013 Elsevier B.V. All rights reserved.

1. Introduction

In the framework of the European Fusion Program, based mainly on magnetic confinement Tokamak-type machines, the complete physical and technological basic demonstration of fusion was approached by the ITER [1] engineering and conceptual design. In particular ITER will be the first challenge to demonstrate licensable fusion safety and environmental potential of fusion and thereby provide a good precedent for the safety of future fusion power reactors [9]. In next generation devices, "dust" will play an important role in determining their safety and operational performance. Because of the nature of its operations, a Tokamak generates aerosol particulate, broken flakes, globules, chunks, and other debris, that may affect its safety and operational performance. In particular in a deuterium–tritium (DT) device, depending on the choice of materials, these products, generically named "dust," may be radioactive from activation and/or T retention, potentially toxic, chemically reactive and explosive. A fraction of the dust inventory can be

mobilized in accident scenarios or during maintenance, threatening the safety of staff and workers on site, the local population and the environment. Possible safety consequences from accidental dust mobilization prompted greater attention to dust in the safety analyses of high energy density machines. In this field, the scientific community has given priority to the issue that concerns the simulation of dust transport caused by a continuum phase, water or air, ingress into the VV. In particular the specific approach includes the development of scaled experiments and numerical models to investigate mobilization and dust's behavior during accidents. Experiments should appropriately simulate geometry, flow conditions, temperature distributions, and structural components that effect mobilization of dust with different characteristics in order to allow the validation and benchmarking of the numerical models. This work is intended to contribute toward improving the understanding of processes taking place during air LOVA, in particular of the velocity field. The simulation of LOVA scenario is a challenging task for today's numerical methods and models because it involves three dimensional geometry with large volume, vacuum multiphase flows ranging from highly supersonic to nearly incompressible and contemporary heat transfer. The present work deals with a single phase large eddy simulation (LES) of the velocity field

* Corresponding author.

E-mail address: malizia@ing.uniroma2.it (A. Malizia).

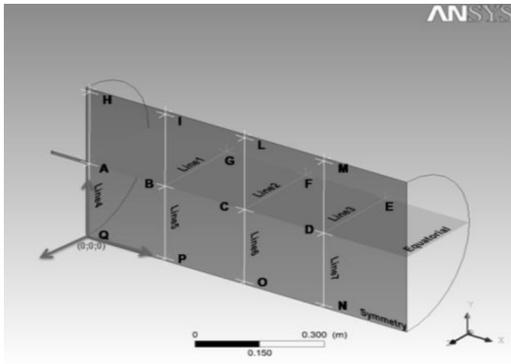


Fig. 1. Positions of the sensors into the tank and computational domain.

during a vessel filling at near vacuum conditions to support the safety study of LOVA events triggered by air income. The model results are compared to the experimental data provided by STARDUST facility at different pressurization rates (100 Pa/s, 300 Pa/s and 500 Pa/s).

2. STARDUST experimental facility

As mentioned above, scaled experiments are considered a priority in order to investigate mobilization and dust's behavior during LOVA. A small tank for aerosol removal and dust (STARDUST) [2–5] has been set up at the ENEA Fusion Technology Department laboratories of Frascati. This facility has been subsequently moved and upgraded by Quantum Electronics and Plasma Physics, a research group located at University of Roma Tor Vergata mainly referring to the acquisition electronics and sensors. The aim of STARDUST development was to experimentally study thermal and flow fields and dust mobilization mechanisms expected in a hypothetical LOVA in ITER. In particular, STARDUST can reproduce a LOVA event due to a small air leakage in ITER VV, with different pressurization rates (100 Pa/s, 300 Pa/s and 500 Pa/s), and for two different positions of the leak, at the equatorial port level and at the divertor port level. This facility also allows the evaluation of obstacles [2] and walls' temperature influence on dust resuspension during both maintenance (MC) and accident (AC) conditions ($T_{\text{wall}} = 25^\circ\text{C}$ for MC, 110°C for AC). In order to measure the pressure time trend inside STARDUST two pressure transducers have been used. The punctual velocity inside the tank has been measured by velocity sensors fixed on a circular support located at different positions (Fig. 1) inside the tank (Fig. 2).

The transducer #455 has been positioned at points A, B, C, and D with the sensible element facing the valves. The pressure transducer #461 has been positioned at the other points by rotating the

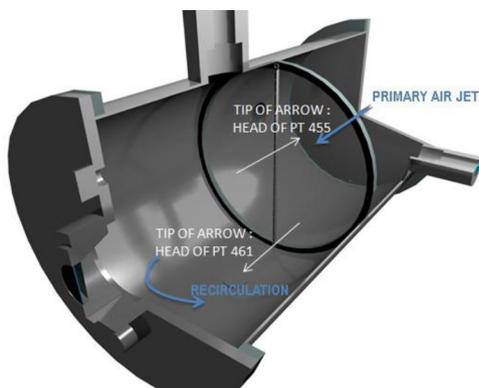


Fig. 2. Pressure transducers orientation inside STARDUST.

support with the sensible element facing the lid of the vessel, in order to measure the velocity component due to recirculation after the impact with the vessel's lid of the primary air jet (Fig. 2).

3. Simulation

The numerical simulations have been carried out to reproduce the flow field for the same scenarios used during the experimental tests in STARDUST; the commercial CFD code ANSYS-CFX [6] has been used. "The CPU time required for a 5 s transient simulation has been about ~45days on the ENEA cluster (512 processors). Assuming the same spatial resolution a full 3D simulation for ITER is practically impossible. Anyway taking into account the toroid symmetry a $15\text{--}20^\circ$ ITER section should be suitable to simulate the pressurization transient with less than 2 million cells and similar CPU time."

3.1. LES methodology

In large eddy simulations [8], an attempt is made to capture the large scale unsteady motions which carry the bulk of the mass, momentum and energy in a flow. The length scale on which these "resolved scale" motions occur depends primarily on the local mesh resolution. The small scale motions that occur on length scales smaller than the mesh spacing cannot be captured and their effects on the resolved scale motions must be modeled. As a small part of the flow has been modeled in LES rather than in RANS calculations, it is hoped that the models will be more accurate as well as simpler to formulate. Whether or not this hope is justified is not clear at the current time. Mathematically, the LES procedure can be thought of as a convolution of the exact turbulent velocity field $\tilde{\mu}$ with a filter function K that gives the resolved scale velocity field $\bar{\mu}$

$$\bar{\mu}(r) = \int K(\Delta|r - r'|)\mu(r')d^3r'$$

The filtering operation is implicit in the formulation (i.e. not explicitly carried out). Under assumptions which are generally non-restrictive, filtering and differentiation commute, i.e. $\partial\bar{\mu}/\partial x = \bar{\mu}\partial/\partial x$. An equation for $\bar{\mu}$ is obtained by convolving the Navier–Stokes equations with the same spatial filter function

$$\frac{\partial\bar{\mu}}{\partial t} + \nabla \cdot \bar{\mu}\bar{\mu} = -\nabla\bar{P} + \nu\nabla^2\bar{\mu}$$

As in conventional turbulence modeling, the nonlinear terms are not closed, because the filtered non-linear terms $\bar{\mu}\bar{\mu}$ cannot be written in terms of the known resolved component $\bar{\mu}$. To overcome this problem, a subgrid-scale stress (SGSS) τ is introduced (in a way similar to RANS modeling), such that $\bar{\mu}\bar{\mu} = \bar{\mu}\bar{\mu} + \tau$. The momentum equation then becomes

$$\frac{\partial\bar{\mu}}{\partial t} + \nabla \cdot \bar{\mu}\bar{\mu} = -\nabla\bar{P} + \nu\nabla^2\bar{\mu} - \nabla \cdot \tau$$

The turbulence modeling task is to estimate the subgrid-scale stress τ from the resolved velocity field $\bar{\mu}$. The simplest SGSS models are of the eddy viscosity type with the Smagorinsky mixing length model, the most well-known; it is used in this study. In eddy viscosity models, it is assumed that the anisotropic part of τ is related to the resolved strain rate field through a scalar eddy viscosity.

$$\tau - \frac{1}{3}\text{tr}(\tau)\delta = -2\nu_t\bar{S} = -\nu_t[\nabla\bar{\mu} + (\nabla\bar{\mu})^t]$$

with the isotropic part of τ being subsumed in P . In the Smagorinsky model, the turbulent eddy viscosity is written as

$$\nu_t = (C_S\nabla)^2\text{tr}(2\bar{S}^2)^{1/2}$$

Table 1
Particles speed in gas inside the tank.

Speed		
Most probable speed	410	m/s
Mean velocity	463	m/s
Mean quadratic velocity	502	m/s
Speed of sound (Laplacian)	346	m/s

where S is the second invariant of the rate-of-strain tensor, C_S is a model constant and $\nabla = (\Delta x \Delta y \Delta z)^{1/3}$ is a measure of the local grid length scale.

3.2. Mesh parameters

A 3D axial-symmetric domain is considered in performing the CFD simulations. For LES, the computational grid must be chosen such that the separation of the resolved and the subgrid-scales occurs in the inertial subrange of the energy spectrum. A further important consideration to resolve LES is the mesh spacing near the wall. According to the recommendations for LES, the first mesh point away from the wall should be located at $y^+ < 1$. The spatial discretization consists of hexahedral elements. A fine grid (2.264.402 cells) has been chosen to be one order of magnitude larger than that of the smallest scale (Kolmogorov scale).

3.3. Boundary conditions

At the inlet surface an appropriate mass flow rate is imposed in order to achieve the different pressurization rates (100 Pa/s, 300 Pa/s and 500 Pa/s) used during the experimental campaign. The inlet air temperature is set to the environmental value ($T = 296$ K). No-slip boundary conditions are assumed at all the walls. A convection heat transfer coefficient is considered for all the walls of the vessel [7]:

$$\overline{Nu} = \left[0.825 \frac{0.387 Ra^{1/6}}{[1 + (0.492/Pr)]^{9/16}} \right]^{8/27}$$

Heat flux at the wall boundary is calculated using $q_w = h_C(T - T_{nw})$, where h_C is a specified heat transfer coefficient (from Nusselt number \overline{Nu} , where Ra is Rayleigh number and Pr is the Prandtl number), T_b is the specified boundary temperature (that is outside the fluid domain) and T_{nw} is the temperature at the internal near-wall boundary element's central node. As internal initial conditions a pressure value of 100 Pa is considered. At this vacuum level the Knudsen number, defined as the ratio of the molecular mean free path's length to a representative physical length scale (inlet diameter), is $\ll 1$. This is very important because traditional continuum CFD techniques are often invalid to analyze gas flows with Knudsen ~ 1 . The operating fluid used in the simulations is dry air assumed initially at rest. The density of the fluid is evaluated using the standard Redlich–Kwong real gas model and the other thermodynamic properties are considered variable according to the internal database. The gas inlet is taken as an experimental mass-flow inlet in order to take into account the time required to open the inlet valve and the time required to reach the regime of the mass flow rate.

At the 100 Pa vacuum level the Knudsen number, defined as the ratio of the molecular mean free path length to a representative physical length scale (inlet diameter), is < 1 (test case conditions of 100 Pa and temperature of 293 K). The Maxwell–Boltzmann distribution that describes particle speeds (Table 1) in gas inside the tank is showed in Fig. 3

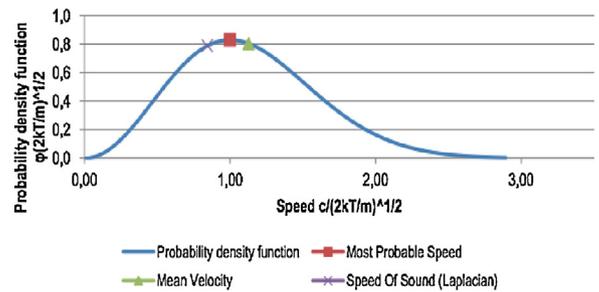


Fig. 3. Maxwell-Boltzmann molecular speed distribution.

4. Results

An analytic solution of pressure history of adiabatic and isothermal pressurization of a confined volume has been adopted [4] in order to allow a comparison between the developed CFD model and a simplified analytical model known in scientific literature [5]. The data extracted by the simulations at the same points where the velocity transducers have been placed during the experimental campaign showed a satisfying agreement with the experimental ones and confirmed the evidences highlighted by the data analysis. In the present section, with a frequency of 2 Hz, inside the STAR-DUST vessel are compared to those provided by the simulations obtained with the model and with the analytic unsteady solution of pressurization process. Numerical and experimental results for 100 Pa test in maintenance condition (MC) at different pressurization rates will be discussed, as are considered the most relevant for the model validation case. The average pressure inside the vessel during the simulations shows a linear trend (Fig. 4) and has been compared with other turbulence models [3]. The model that approaches better the experimental trend is the one that uses the LES turbulence model. The axial velocity (Fig. 5) in points A and D (see Fig. 1) evaluated close to the air injection, increases from zero to the maximum in the first second and then decreases. The CFD applications show stable numerical results in modeling local gas velocity field at low pressure conditions. Velocities remain very high for all transient. Assuming that the axial velocity profiles are symmetric with respect to the symmetry plane, the velocity's direction falls relatively rapidly with distance from the injection section along the axis (from 1/4 to 3/4 of the vessel about 25%). The simulation shows a satisfying global agreement with experimental data for all performed experiments and simulation (from 1/3 to 2/3 of the vessel about 33%). The high axial velocity region close to the air inlet corresponds to the expansion in supersonic regime.

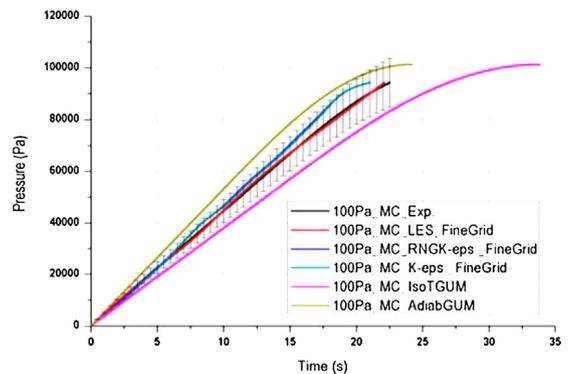


Fig. 4. Average pressure time trend for different turbulence (LES, RNG k -epsilon, standard k -epsilon) model (internal pressure = 100 Pa, $T_{wall} = 25$ °C for MC) and analytic solution (IsoTGUM = isothermal, AdiabTGUM = adiabatic).

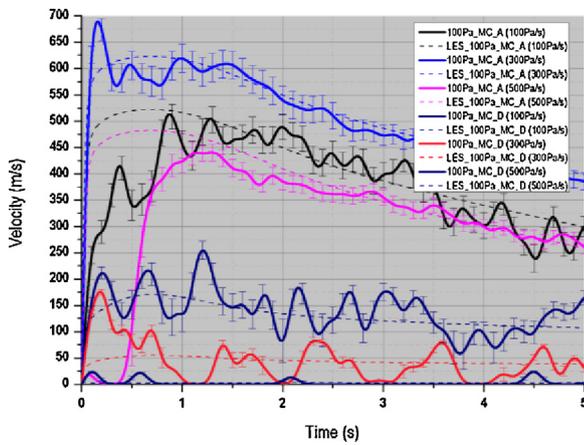


Fig. 5. Axial velocity at points A and D inside the vessel (100 Pa, 100–300–500 Pa/s, MC).

Observing the experimental velocity profile inside the vessel we can see a reduction of axial velocity at points for 500 Pa/s pressurization rate case (Fig. 6). This is caused by the typical compressible flow effect named choked effect. In the case of high upstream air pressure and vacuum conditions downstream of an inlet section (orifice), both the air velocity and the mass flow rate become limited when sonic velocity is reached through the inlet section and the expansion inside the vessel becomes limited too. This means that the highest velocities, the driver parameter for dust mobilization inside the vessel, are reached for low pressurization rates, about 300 Pa/s; that evidence leads to an increase in the attention paid to the dust mobilization problem in the field of the safety research, as those conditions in which the highest velocities are reached are the ones the IAEA evaluated the most common [1].

From the streamlines analysis (Fig. 7), after vacuum breakdown, the jet generates vortex structures that roll up several times, and the inlet gas particles follow the larger one which brushes and interacts with all boundaries.

It has relevant safety implication because the mobilized dust, that follows the streamline during the motion, can escape or be trapped in VV asperity and then accumulated. This high dust concentration zones are able to raise the probability of chemical reaction and in particular to increase the reactive surface area that may contribute to hydrogen production and potential explosion.

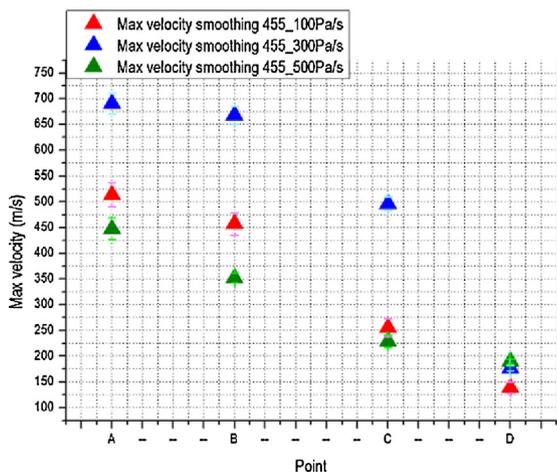


Fig. 6. Maximum velocities comparison at different pressurization rates (100, 300 and 500 Pa/s, MC).

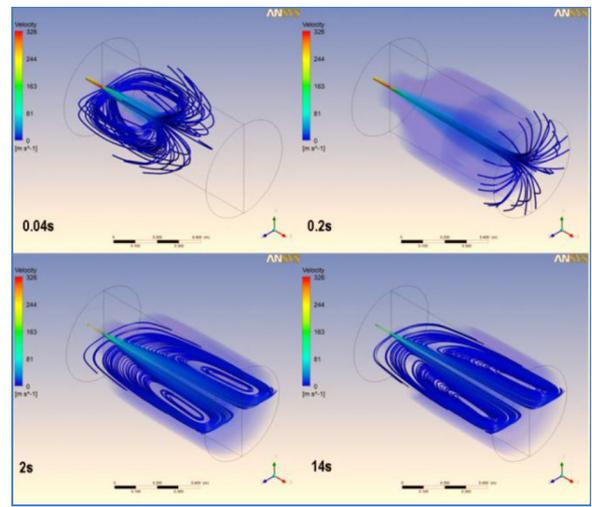


Fig. 7. Velocity streamlines inside STARDUST.

5. Conclusion

The aim of this work has been to contribute toward improving the understanding of processes taking place during a LOVA event. The CFD simulations and experimental activities in STARDUST have been carried out in strong correlation in order both to understand the capabilities of computational codes and to predict correctly the characteristics of the flows during a LOVA. The LES computations show a better agreement with the experiments than RANS in modeling local gas velocity field at low pressure conditions. The influence on the air velocity of the pressurization rates is much more remarkable when the maximum is reached at 300 Pa/s. A detailed experimental analysis will be necessary. The model has shown very promising results, especially in terms of accuracy and quality of analysis. Of course other improvements have to be done in order to better characterize and develop the multiphase model of LOVA here presented. This work can be seen as a starting point. Many aspects need to be verified and others need to be investigated more deeply.

References

- [1] IAEA, ITER EDA Documentation Series No. 24, s.n., ITER Technical Basis, Vienna, 2002.
- [2] C. Bellecci, P. Gaudio, I. Lupelli, A. Malizia, M.T. Porfiri, R. Quaranta, M. Richetta, STARDUST experimental campaign and numerical simulations: influence of obstacles and temperature on dust resuspension in a vacuum vessel under LOVA, Nuclear Fusion 51 (2011) 053017, Nuclear Fusion, IOPscience, IOP Publishing and International Atomic Energy Agency.
- [3] P. Gaudio, A. Malizia, I. Lupelli, RNG *k-ε* modelling and mobilization experiments of loss of vacuum in small tanks for nuclear fusion safety applications, International Journal of Systems Engineering, Applications and Development 5 (287) (2011) 287–305, ISSN: 2074-1308.
- [4] P. Gaudio, A. Malizia, I. Lupelli, Experimental and numerical analysis of dust resuspension for supporting chemical and radiological risk assessment in a nuclear fusion device, in: International Conference on Mathematical Models for Engineering Science (MMES'10), Puerto De La Cruz, Tenerife, November 30–December 2, 2010, pp. 134–147.
- [5] C. Bellecci, P. Gaudio, I. Lupelli, M.T. Malizia, I. Porfiri, R. Quaranta, M. Richetta, CFD analysis of loss of vacuum accident for safety application in experimental fusion facility, in: European COMSOL Conference, Milano, October 14–16, 2009, ISBN/ISSN: 978-0-9825697-2-6.
- [6] ANSYS, ANSYS CFX-13 User Guide.
- [7] Theodore L. Bergman, David P. Dewitt, David P. DeWitt, Frank P. Incropera and Adrienne S. Lavine, Fundamentals of Heat and Mass Transfer, 2011, Hardcover, ISBN-10: 0470501979. ISBN-13: 9780470501979.
- [8] D.C. Wilcox, Turbulence Modeling for CFD, DCW Industries, La Canada, CA, 1994.
- [9] T. Pinna, et al., Operating experiences from existing fusion facilities in view of ITER safety and reliability, Fusion Engineering and Design, Elsevier, Fusion Engineering and Design 85 (2010) 1410–1415.