Fusion Engineering and Design 84 (2009) 1479–1483

Contents lists available at ScienceDirect

Fusion Engineering and Design

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

Hydraulics and heat transfer in the IFMIF liquid lithium target: CFD calculations

A. Peña[a,*], G.A. Esteban[a], J. Sancho[a], V. Kolesnik[b], A. Abánades[c]

[a] University of the Basque Country, Nuclear Engineering and Fluid Mechanics Department, Alameda de Urquijo s/n, 48013 Bilbao, Spain
[b] ZAO "EMT-R" 46, Suzdalskaya Street, 111672 Moscow, Russia
[c] Polytechnic University of Madrid, Power Engineering and Fluid Mechanics Department, José Gutiérrez Abascal 2, 28006 Madrid, Spain

ARTICLE INFO

Article history:
Available online 5 April 2009

Keywords:
CFD
IFMIF
Free surface
Liquid metal
Thermal hydraulics
Boiling point

ABSTRACT

CFD (Computational fluid dynamics) calculation turns out to be a good approximation to the real behavior of the lithium (Li) flow of the target of the international fusion materials irradiation facility (IFMIF). A three-dimensional (3D) modelling of the IFMIF design Li target assembly, made with the CFD commercial code ANSYS-FLUENT has been carried out. The simulation by a structural mesh is focused on the thermal-hydraulic analysis inside the Li jet flow. For, this purpose, the two deuteron beams energy deposition profile is modelled as an energy source term inside the volume of liquid affected. Turbulence is estimated using the RNG k–ε model, and a surface-tracking technique applied to a fixed Eulerian mesh called volume of fluid (VOF) is used to determine the position of the free surface. Calculations varying the jet velocity from a range of 10–20 m/s, show that maximum calculated temperatures are still below the lithium's boiling point, due to the increase of the pressure induced by centrifugal force.

1. Introduction

The international fusion materials irradiation facility (IFMIF) is currently being planned by Japan, Russia, the EU, and the United States [1], and it is an accelerator-based deuteron–lithium neutron source thought to produce intense high energy neutrons for testing fusion materials to be used in ITER and fusion DEMO reactor. Two deuteron beams at an energy of 40 MeV (2×125 mA) will be injected into a high-speed liquid lithium (Li) jet, flowing along a vertical concave wall in a 10^{-3} Pa vacuum. An average surface heat flux of 1 GW/m^2 on the Li free surface will be produced by the irradiation of the beam.

The IFMIF Li target assembly is designed to provide a stable Li jet in a safe operation mode. Great waves could have effects on the neutron field, and there could be a possibility of wall burning by the beams. On the other hand, high velocities up to 20 m/s are needed to remove the deposited beam energy. Table 1 shows the IFMIF main specifications to be taken into account for the numerical simulation.

In Fig. 1, a scheme of the IFMIF design allows a general description of the main features involved in the target flow: the concave back-wall, the Li free surface, the vacuum zone and the beam impact zone.

The concave back wall, whose radius is 25 cm avoids Li boiling because there is a centrifugal induced overpressure [2].

The simulation of working conditions with the available engineering computational fluid dynamic (CFD) codes is part of the design work, and the validation of such codes will always be the proper way to improve them and to rely on their predicted results.

This study shows calculations made with the CFD commercial code ANSYS-FLUENT [3], focused on the thermal-hydraulic analysis of technology issues such as a vacuum environment, a high heat energy source, a liquid metal flow, and a free surface flow. Considerations are made about the capabilities and deficiencies of the models included in ANSYS-FLUENT when dealing with the mentioned physical phenomena.

2. Thermal-hydraulic analysis with ANSYS-FLUENT CFD code

2.1. Grid and physical models

The geometry grid for the IFMIF target simulation is shown in Fig. 2. A meshing of 881,160 hexahedral cells has been used for the simulation. The grid density of the region between the concave wall and up to 25 mm from it, is 20 elements in perpendicular direction from the wall. Therefore, cells are 1.25 mm high in that direction. The first cell next to this zone is 3.8 mm high. This cell structure is maintained down to the outlet.

A two phase flow problem has to be solved: Lithium and a vacuum environment. This last one being air at 10^{-3} Pa. One of the main concerns about this flow is the stability of the liquid metal free surface. To track this surface, CFD codes as ANSYS-FLUENT use a surface-tracking technique applied to a fixed Eulerian mesh

* Corresponding author. Tel.: +34946014275; fax: +34946014159.
E-mail address: alberto.bandres@ehu.es (A. Peña).

0920-3796/$ – see front matter © 2009 Elsevier B.V. All rights reserved.
doi:10.1016/j.fusengdes.2009.03.001

Table 1
IFMIF target specification.

Area of the Li Jet (width × thickness)	260 mm × 25 mm
Velocities of the Li jet	A range from 10–20 m/s
Inlet temperature of Li	250 °C
Pressure at free surface (vacuum)	10^{-3} Pa
Energy (beam) deposition surface	200 (width) × 50 mm (height)

called volume of fluid (VOF). It is designed for two or more immiscible fluids where the position of the interface between the fluids is of interest. This is the case, as a Lithium jet flows in a vacuum environment along a concave wall.

The volume fraction for phase q (α_q)is calculated through the transport Eq. (1) [3]:

$$\frac{\partial(\alpha_q \rho_q)}{\partial t} + \nabla(\alpha_q \rho_q \vec{v}) = S_{\alpha_q} + \sum_{p=1}^{n}(\dot{m}_{pq} - \dot{m}_{qp}) \quad (1)$$

where S_{α_q} is the source term for the volume fraction for phase q. By default this source term is zero. $\dot{m}_{qp}$ is the mass transfer from phase q to phase p, and $\dot{m}_{pq}$ is the mass transfer from phase p to phase q.

The volume fraction for the other phases must follow the expression (2):

$$\sum_{q=1}^{n} \alpha_q = 1 \quad (2)$$

The material properties, φ, in one computational cell are calculated as average values of each phase properties, using Eq. (3):

$$\phi = \sum \alpha_q \phi_q \quad (3)$$

Therefore, if there is only one phase in that cell, the property will correspond to that phase. Discretization of the transport equation has been done with an explicit time scheme. For this scheme the default Courant number is 0.25, and it is the value used for the calculations. The time step for VOF calculation is refined, based on this maximum Courant number allowed near the free surface. It compares the time step in the calculation with the characteristic time of transit of a fluid element across a control volume. This time is calculated, in the region near the fluid interface, dividing the volume of each cell by the sum of the outgoing fluxes. The convergence criterion for the transport equation is also the default one, 10^{-3}.

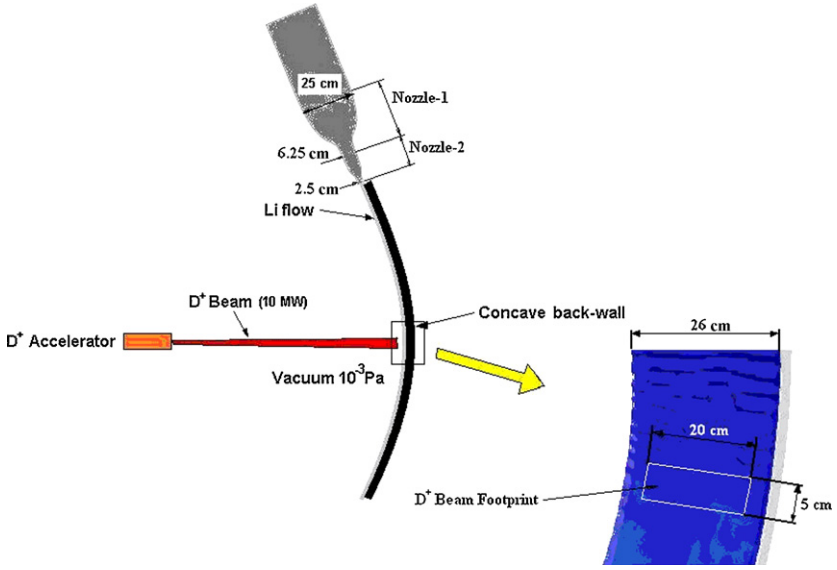

Fig. 1. Scheme of the IFMIF target geometry.

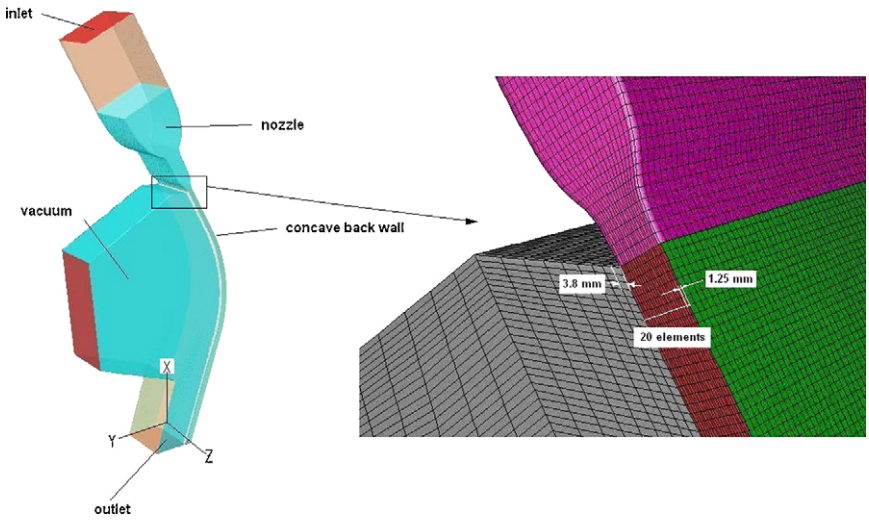

Fig. 2. IFMIF target geometry.