DSMC analysis of a Loss-of-Coolant Accident from a He-cooled blanket in the EU DEMO Vacuum Vessel

Introduction

The Loss-Of-Coolant Accident (LOCA) inside the Vacuum Vessel (VV) of the EU DEMO fusion reactor is of interest for the design of the tokamak, since the peak pressure induced by the accident in the VV should stay below a prescribed threshold. In order to quantify the pressure peaks on the VV walls during the accident, a detailed analysis is needed. In particular, a 3D Computational Fluid Dynamics (CFD) transient analysis of the hypersonic ($Ma \gg 1$) flow developed during the accident is needed. This analysis is currently carried out on the Helium Cooled Pebble Bed (HCPB) [1] blanket design.

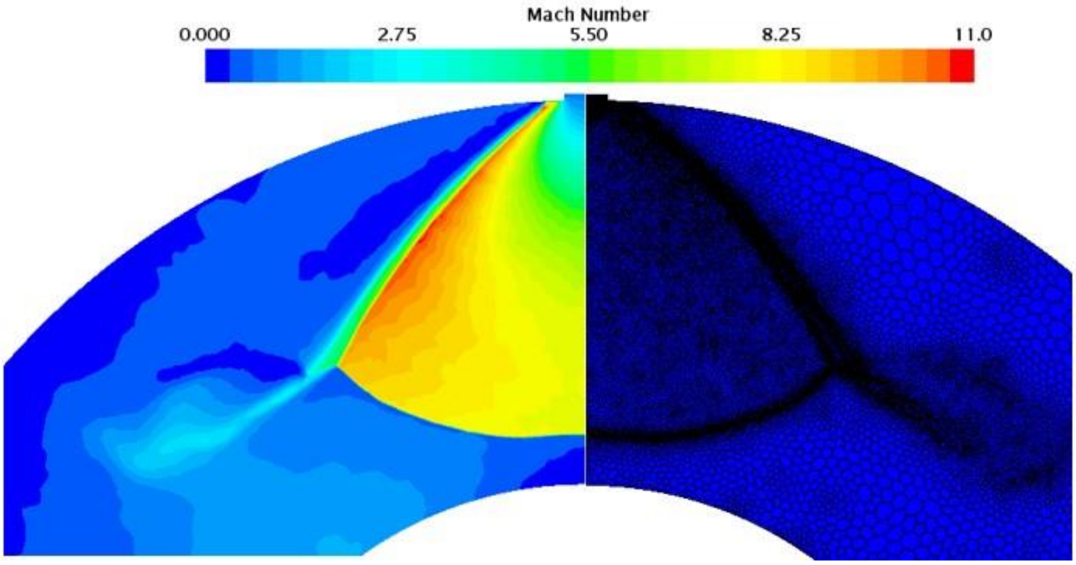

Figure 1: (left) Preliminary computation of the Mach number during a Helium LOCA in the vacuum vessel and (right) the adaptive mesh adopted to follow the fronts of the Helium jet.

Aim of the work

The in-Vessel LOCA is characterized by the expansion of pressurized He towards high vacuum. In the initial phase of the transient -i.e. after the rupture- the expanding gas is highly rarefied. This has been proven in [2] based on the evaluation of the Knudsen number Kn, a dimensionless parameter defined as the ratio between the mean free path of a particle and a characteristic length of the system. This means that the continuum assumption is not applicable to this initial phase, which therefore cannot be studied via conventional CFD codes. In other words, a kinetic model is required, as a fluid model is not applicable.

The Direct Simulation Monte Carlo (DSMC) method is a promising candidate to model the first instants of the in-vessel LOCA, as it provides an efficient solution to the kinetic Boltzmann equation for medium to large values of Kn [3]. Recently, the DSMC method has been included in OpenFOAM, an open-source toolbox for the computational solution of flow problems [4]. This DSMC implementation has already been employed at Politecnico di Torino to solve other kind of rarefied flow problems [5] and can exploit High-Performance Computing (HPC).

The aim of this proposal is to employ OpenFOAM to simulate the evolution of the initial transient, i.e. up to the point where sufficient collisionality is reached and CFD models can be applied.

Keywords: HPC, rarefied flow, DSMC, open source, hypersonic flow

Contact persons

- Dr. Antonio Froio: antonio.froio@polito.it
- Ing. Giuseppe Francesco Nallo: giuseppefrancesco.nallo@polito.it
- Andrea Zappatore: andrea.zappatore@polito.it

References

[1] F. A. Hernández, et al., "Overview of the HCPB Research Activities in EUROfusion," IEEE Transactions on Plasma Science 46(6):2247-2261, 2018.

[2] G. Digioia, CFD analysis of a Loss-of-Coolant Accident from a He-cooled blanket in the EU DEMO Vacuum Vessel, M.Sc. thesis, Politecnico di Torino, 2018

[3] G. A. Bird, *Molecular Gas Dynamics and the Direct Simulation of Gas Flows*. Oxford: Clarendon Press, 1994.

[4] C. White *et al.*, "DsmcFoam+: An OpenFOAM based direct simulation Monte Carlo solver," *Comput. Phys. Commun.*, vol. 224, pp. 22–43, 2017.

[5] G. F. Nallo, G. Mazzitelli, L. Savoldi, F. Subba, and R. Zanino, "Self-consistent modelling of a liquid metal box-type divertor with application to the divertor tokamak test facility: Li versus Sn," *Nucl. Fusion*, vol. 59, no. 6, 2019