Title: Development and validation of magneto-hydrodynamic turbulence models for the thermal-hydraulic design of ARC-class fusion reactor liquid blankets

Authors: M. Caravello[a,*], N. Abrate[a], A. Aimetta[a], E. Baglietto[b], S. Dulla[a], A. Froio[a], N. Mancini[c], F. Podenzani[c], R. Zanino[a,&]

[a]*NEMO group, Dipartimento Energia, Politecnico di Torino, 10129 Torino, Italy*
[b]*Department of Nuclear Science and Engineering, Massachusetts Institute of Technology, Cambridge, MA 02139, USA*
[c]*Research & Technological Innovation Department, Eni S.p.A., 20097 San Donato Milanese (MI), Italy*
**Corresponding author: marco.caravello@polito.it*
[&]Presenting author: roberto.zanino@polito.it

Abstract This work presents the implementation of turbulence models for analyzing the thermal-hydraulic performance of liquid blankets in ARC-class fusion reactors. Implemented within the OpenFOAM framework, these models include magnetohydrodynamic (MHD) modification of the standard Reynolds-Averaged Navier-Stokes (RANS) closure models, including k-epsilon and Reynolds Stress Model. The MHD models can be integrated into nemoFoam, a previously developed multi-physics modular tool, to enable more accurate simulations of fusion reactor blankets. The nemoFoam code couples computational fluid dynamics (CFD) with simplified neutron transport models, offering a comprehensive platform for the thermal-hydraulic and neutronic optimization of fusion reactor blankets. The coupling is achieved through iterative exchange of power density distribution and thermal field between the neutronics and the thermal-hydraulic module. The MHD modeling capabilities have been validated against experimental data while neutronic performance has been already cross-verified, in previous works, through code-to-code benchmarks with Monte Carlo Serpent simulations.

Introduction Some design of nuclear fusion reactors, such as the ARC-class reactors currently under development at Commonwealth Fusion System (CFS), aim to deliver affordable and robust solutions for sustainable energy production using a molten salt (MS) blanket, which has to perform a number of critical functions efficiently and simultaneously such as tritium breeding, heat removal and neutronic shielding. However, the use of a molten salt coolant introduces significant challenges in thermal-hydraulic modeling due to its high viscosity and large Prandtl number, which affects fluid dynamics and heat transport characteristics. Additionally, since the coolant is an electrically conductive fluid circulating in a region with magnetic fields, electromagnetic forces (Lorentz forces) arise in the liquid regions. These forces distort the classical velocity field, altering turbulence structures and inducing flow anisotropy. The flow field directly influences the distribution of the electric currents induced in the MS by Lorentz forces, which further impacts its behavior. This study addresses this challenges by implementing modified turbulence models to capture these effects and proposes comparison between simulations and experimental results to quantify improvements in simulation accuracy. This works aims to propose a tool useful for designing safer and more efficient MS liquid blankets, as in ARC-class fusion reactors. With the ability to correctly model the thermal-hydraulic behavior of MS, it becomes possible to assess other phenomena that depend on the velocity field and the discharge process of the induced electric currents. These phenomena may include corrosion and excitation of structural materials in the blanket, which are crucial to understanding the overall behavior and longevity of nuclear fusion reactor designs.

Methodology OpenFOAM, an open-source computational fluid dynamics (CFD) platform, is used for model development and simulation. Key advancements include:

1. **Implementation of Turbulence Models**: The k-ε and Reynolds Stress Models (RSM) were enhanced with MHD-specific formulations, integrating the effects of the electromagnetic field on turbulent variables. An alternative approach, as the addition of a scalar transport equation for the anisotropy variable α, was implemented according to the Widlund model. The introduction of α accounts for the effect of the magnetic field on turbulence isotropy, reducing turbulence intensity in the direction parallel to the magnetic field and increasing it in the perpendicular direction. The turbulence models gain information regarding the dimensionality of the turbulence, helping determine whether the turbulence is more 2D or 3D.
2. **Model Validation**: The models were validated using literature cases and experimental data, focusing on flow behavior over the range (10,000-30,000) of Reynolds number and the range (0-3000) of Hartmann number.
3. **Integration with Multiphysics Workflows**: Coupling CFD with simplified neutron transport provided fast insights into the interplay between thermal hydraulics and neutronics. The nemoFoam (Figure 1) code uses transient neutron diffusion multi-group equations and photon mono-kinetic diffusion equation. After calculating neutron and photon distributions, the power deposition field is determined using KERMA coefficients.

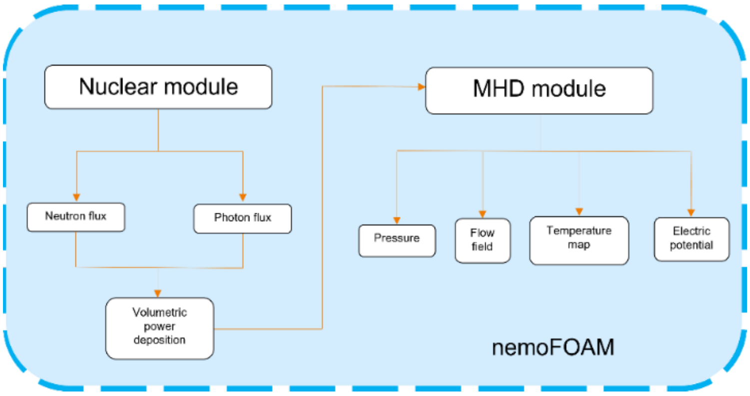

Figure 1: Schematic of the multi-physics model integration in the nemoFoam code

Results The study achieved several goals:

1. **MHD Model Benchmarking and Validation**: The MHD turbulence models were benchmarked against reference cases, confirming their applicability to ARC-class blanket designs. A comparison of velocity profiles from simulations and experimental data (Figure 2) highlights the improved agreement achieved by the MHD k-epsilon model. Specifically, the normalized radial velocity profile as a function of the normalized radius shows better accuracy near the wall, closely matching experimental measurements from [Takeuichi-2007].
2. **Multi-physics Workflow Optimization**: The integration of different physical phenomena in a modular tool relying on CAD-based neutron transport simulations and CFD, could facilitate comprehensive analyses of blanket performance, including neutron flux distribution, volumetric power deposition (Figure 3), tritium production, flow patterns and temperature maps.

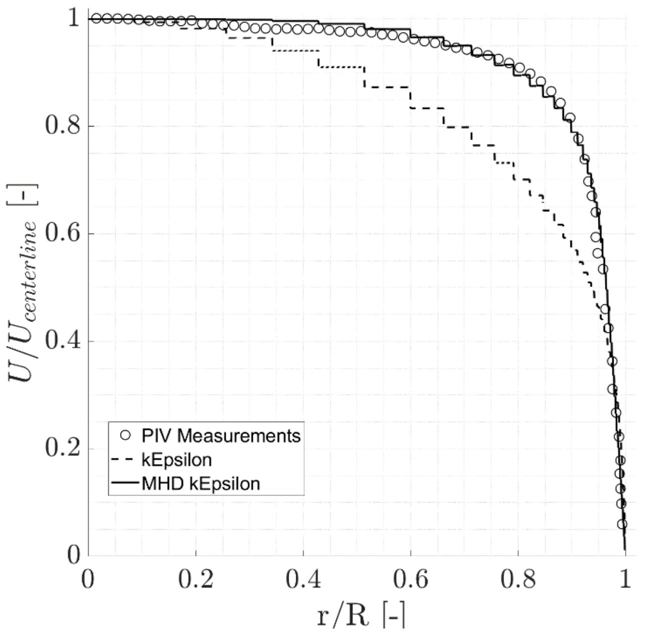

Figure 2: Comparison of velocity profiles from simulations and experimental data.

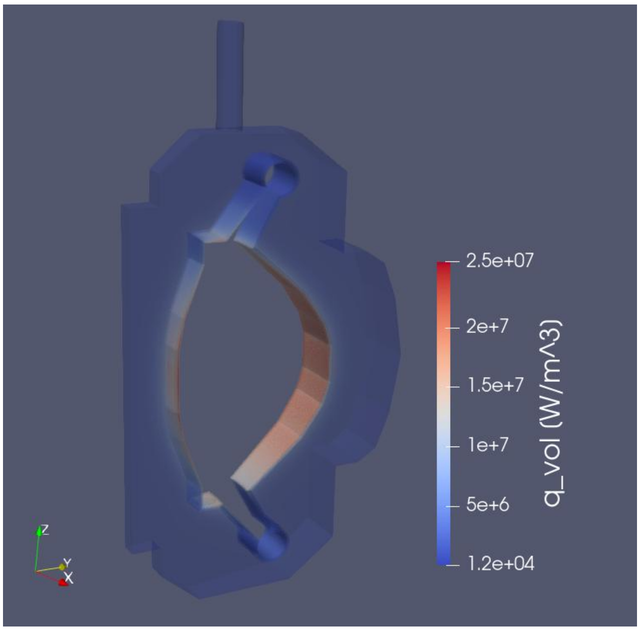

Figure 3: Power deposition profile in the reactor blanket.

Conclusions and perspective This work contributes to the development and validation of open-source computational tools to support the design and optimization of innovative nuclear fusion energy systems. The implementation and validation of advanced turbulence models not only enhance our understanding of liquid blanket behavior but also provide a foundation for future research and design optimization. Future efforts include extending the models to cover broader operational scenarios, refining predictions of TBR, and integrating them with structural and material analysis tools to address corrosion and material degradation.

Acknowledgements: This work is part of the project PNRR-NGEU which received funding from the Italian Ministry of University and Research – DM 352/2022