POLITECNICO DI TORINO
Repository ISTITUZIONALE

Development and validation of magneto-hydrodynamics turbulence models for the thermal-hydraulic design of fusion reactor liquid blankets

Original
Development and validation of magneto-hydrodynamics turbulence models for the thermal-hydraulic design of fusion reactor liquid blankets / Caravello, M., Abrate, N., Aimetta, A., Dulla, S., Froio, A., Zanino, R., Mancini, N., Podenzani, F., Baglietto, E.. - ELETTRONICO. - (2025). (Fusion Energy Conference 2025 Chengdu (CN) 13-18/10/2025).

Availability:
This version is available at: 11583/3004701 since: 2025-10-31T18:45:45Z

Publisher:
IAEA

Published
DOI:

Terms of use:

This article is made available under terms and conditions as specified in the corresponding bibliographic description in the repository

Publisher copyright

(Article begins on next page)

18 November 2026

CONFERENCE PRE-PRINT

DEVELOPMENT AND VALIDATION OF MAGNETO-HYDRODYNAMICS TURBULENCE MODELS FOR THE THERMAL-HYDRAULIC DESIGN OF FUSION REACTOR LIQUID BLANKETS

M. CARAVELLO, N. ABRATE, A. AIMETTA, S. DULLA, A. FROIO, R. ZANINO
NEMO group, Dipartimento Energia, Politecnico di Torino
Torino, Italy
Email: marco.caravello@polito.it

N. MANCINI, F. PODENZANI
Research & Technological Innovation Department, Eni S.p.A.,
San Donato Milanese (MI), Italy

E. BAGLIETTO
Department of Nuclear Science and Engineering, Massachusetts Institute of Technology
Cambridge, MA 02139, USA

Abstract

This study presents the application of MHD turbulence models for analysing the thermal-hydraulic performance of liquid blankets in ARC-class fusion reactors. These models have been implemented and tested within an OpenFOAM-based modular multiphysics workflow that couples CFD with simplified neutron transport, enabling efficient evaluation of the volumetric heating, flow redistribution, and thermal-hydraulic performance. Multiphysics simulations on a 3D sector of the ARC blanket are ongoing with the objective to highlight effects of magnetic damping on velocity profiles, pressure drop, and temperature distribution. Preliminary results demonstrate the potential of the OpenFOAM-based platform as a flexible, high-fidelity tool for multiphysics design of fusion reactors blankets.

1. INTRODUCTION

The design of liquid blankets for nuclear fusion reactors represents a multiphysics challenge, requiring a coupled treatment of thermal-hydraulics, neutronics and magnetohydrodynamics (MHD). These physics are tightly connected: volumetric heating from neutron interactions governs fluid flow and temperature distribution, while strong magnetic fields suppress turbulence and alter transport properties, ultimately impacting tritium breeding performance. Addressing such coupled effects within a single framework is essential to provide reliable design tools for fusion reactors. In this context, ARC-class reactors represent a particularly compelling case study. The ARC design [1] aims to demonstrate a compact, affordable, and robust pilot fusion power plant by exploiting high-temperature superconductors and a liquid FLiBe blanket. The liquid blanket simultaneously acts as moderator, coolant, shield, and tritium breeder, making its accurate modelling critical for reactor viability. The present work focuses on a simplified ARC blanket geometry, providing a representative and computationally efficient configuration for the application of multiphysics simulation tools. Experimental data that could calibrate reduced-order or system-level models of such environments are extremely limited due to the high cost and complexity of reproducing blanket-relevant flow conditions under strong magnetic fields. This limitation motivates the development of high-fidelity numerical tools capable of directly capturing the multiphysics interactions that characterize blanket operation. Among the physical effects, MHD plays a relevant role in determining the blanket thermal-hydraulic performance: the interaction between the electrically conducting molten salt and intense magnetic fields leads to turbulence suppression, anisotropic transport, and the formation of complex flow structures, which must be accurately modelled to ensure efficient blanket operation. Several efforts in the nuclear community have tried to build multiphysics environments. For instance, the FERMI project [2] at Oak Ridge National Laboratory integrates MCNP for neutronics, OpenFOAM for thermal-hydraulics, and Diablo for structural mechanics through the preCICE library. In the fission domain, GenFoam [3] leverages OpenFOAM for coupled neutronics, thermal-hydraulics, and thermomechanics simulations, while the MOOSE [4] framework provides a modular finite-element environment widely used across nuclear applications. These examples reflect the growing demand for integrated codes that can consistently handle reactor-relevant multiphysics problems. Within this context, the present work contributes to the development of an open source multiphysics platform based on OpenFOAM, capable of simultaneously addressing advanced MHD turbulence models and simplified neutron transport. The focus is on both implementing and validating turbulence models,

including MHD-adapted k–ε, [5], Reynolds Stress Models (RSM) [6], and Widlund's anisotropy closure [7], while establishing a flexible and extensible code infrastructure. This platform serves as a foundation for progressively integrating additional physics modules, offering another option toward comprehensive design tools for ARC-class blankets.

2. METHODOLOGY

The present work employs OpenFOAM, an open-source CFD platform, as the core tool, which offers flexibility, transparency, and strong community support for modelling complex multi-physics phenomena.

The baseline for the thermal-hydraulic solver used during the study is the standard chtMultiRegionFoam. To incorporate MHD effects, the momentum equation is extended to include the Lorentz force term, so that the momentum equation become the following:

$$\rho\left[\frac{\partial}{\partial t}(\vec{v}) + (\vec{v}\cdot\vec{\nabla})\,\vec{v}\right] = -\vec{\nabla}(p) + \mu_{eff}\,\nabla^2(\vec{v}) + \vec{j}x\vec{B} \tag{1}$$

where j is the induced current density and B the magnetic field. In the case in which the additional magnetic field induced by the MHD effects in small, compared with the external one, it is the case of low magnetic Reynolds number flow. This situation is verified in the ARC-class reactor because the condition $Re_m = \mu\sigma LU \ll 1$ is satisfied, and the problem can be treated as inductionless. The simplification allows to solve a Poisson equation for the electric potential, rather than for the magnetic field and reconstruct the current density directly from the other variables of the problem.

$$\nabla^2(\phi) = \vec{\nabla}\cdot\left[\vec{v}\,x\,\vec{B}\right] \tag{2}$$

$$\vec{j} = \sigma\left[\vec{v}\,x\,\vec{B} - \vec{\nabla}(\phi)\right] \tag{3}$$

Additionally, we can introduce one more simplification of the model introducing the wall conductance ratio, a dimensionless parameter defined as

$$c_w = \frac{\Delta_{wall}\,\sigma_{wall}}{L_{fluid}\,\sigma_{fluid}} \tag{4}$$

Where σ_{wall} and σ_{fluid} are the electrical conductivities of the wall and fluid, respectively, L is a characteristic length of the fluid domain and Δ_{wall} is the wall thickness. This ratio quantifies the relative ability of the wall to carry induced currents compared to the fluid. Low values of the wall conductivity characterize poorly conducting walls, while very good conductive walls present values of c_w well above the unity.
The turbulent behaviour of electrically conducting molten salts under strong magnetic fields presents unique challenges. Standard turbulence closures are insufficient to capture turbulence suppression. To address this, two primary turbulence models were extended:

- MHD-extended k–ε model: The baseline RANS closure adopted is the low-Reynolds anisotropic k–ε model of Baglietto and Ninokata [7]. The original model exploits the nonlinear formulation of the strain rate to correctly calculate the anisotropic flow path of the coolant close to the wall and in proximity of bends and curves. The baseline model was modified to include additional source terms accounting for electromagnetic damping $S_{k,mhd}$ and $S_{\varepsilon,mhd}$, according to the approach proposed by Meng [7]. The dissipation rate ε and turbulent kinetic energy k are directly influenced by the Lorentz force, leading to reduced turbulence intensity along the magnetic field lines.

$$S_{k,mhd} = -\frac{\sigma}{\rho}B^2 k\,exp\left(-C_1^M\sqrt{\frac{\sigma}{\rho}B^2\frac{\nu}{k}}\right) \tag{5}$$

$$S_{\varepsilon,mhd} = -\frac{\sigma}{\rho}B^2\varepsilon\, exp\left(-C_1^M\sqrt{\frac{\sigma}{\rho}B^2\frac{\nu}{k}}\right) \tag{6}$$

- MHD-extended Reynolds Stress Model (RSM): This higher-fidelity closure explicitly solves transport equations for each component of the Reynolds stress tensor. MHD modifications $S_{R_{ij},mhd}$ were incorporated to capture directional suppression of turbulence, transposing the approach proposed by Meng to the standard RSM proposed by Launder, Reece and Rodi.

$$S_{R_{ij},mhd} = -\frac{\sigma}{\rho}B^2R_{ij}\, exp\left(-C_1^M\sqrt{\frac{\sigma}{\rho}B^2\frac{\nu}{k}}\right) \tag{7}$$

- The Widlund anisotropy model was implemented by introducing a scalar transport equation for the anisotropy variable α, used to construct the electromagnetic damping terms $S_{k,mhd}$ and $S_{\varepsilon,mhd}$. The variable α is the representation of the anisotropic nature of the turbulent length scales parallel and perpendicular to the magnetic field, that according to the author should allow the accurate calculation of the Joule dissipation. For the full analytical treatment one can refer to [7].

$$S_{k,mhd} = -\frac{2\sigma}{\rho}B^2\alpha k \tag{8}$$

$$S_{\varepsilon,mhd} = -\frac{2\sigma}{\rho}B^2\alpha\varepsilon \tag{9}$$

The neutronic module implements the neutron diffusion multigroup equation simplified for non-fissile media

$$\frac{1}{v_g}\frac{\partial}{\partial t}(\phi_g) = \vec{\nabla}\cdot\left(\mathrm{D_g}\vec{\nabla}(\phi_g)\right) - \Sigma_g^{abs}\,\phi_g - \sum_{g'=1}^{G,\, g\neq g'}\Sigma_{g\to g'}^{scat}\,\phi_g + \sum_{g'=1}^{G,\, g\neq g'}\Sigma_{g'\to g}^{scat}\,\phi_{g'} + S_g \tag{10}$$

where g indicates the g-th energy group, and G represents the overall number of energy groups, Φ is the neutron scalar flux, D is the diffusion coefficient, $\Sigma_{g\to g'}^{scat}$ is the scattering cross section from group g to group g', Σ_g^{abs} is the absorption cross section and S represents an additional source of neutrons. It has been shown [10] that is mandatory to consider the power deposited also by photons to accurately model the fusion nuclear reactors, in the model it has been implemented also the mono-kinetic diffusion equation for photons:

$$\frac{1}{v}\frac{\partial}{\partial t}(\Phi_\gamma) = \vec{\nabla}\cdot\left(\mathrm{D}_\gamma\vec{\nabla}(\Phi_\gamma)\right) - \Sigma_\gamma^{abs}\Phi_\gamma - \mu_\gamma^{abs}\Phi_\gamma + \Sigma_n^{tot}\nu_\gamma\Phi_n + S_\gamma \tag{11}$$

where γ indicates the photon-related term, D_γ is the P-1 approximation photon diffusion coefficient, Σ_γ^{abs} is the absorption attenuation coefficient and S_γ represents an additional source of photons. The fluxes obtained are used to compute the power deposition fields by means of the Kinetic Energy Released in Matter (KERMA) coefficients:

$$q_{vol} = K_\gamma\Phi_\gamma + \sum_{g'=1}^{G}\mathrm{K}_{g'}\,\phi_{g'} \tag{12}$$

The volumetric power deposition q_{vol} serves as source term in the energy equation of the CFD-MHD module. Compared to a full CFD–Monte Carlo coupling, the diffusion-based solver offers a substantial reduction in computational cost, while preserving the ability to capture the essential physics, as shown in [11].

Although the neutronic and thermal-hydraulic solvers typically operate on the same discretized domain to ensure consistency between physics modules, the framework also allows the use of two separate meshes. This flexibility enables a coarser mesh for neutron calculations and a finer mesh for CFD, reducing computational costs while maintaining accurate coupling.

3. SIMULATION SETUP AND RESULTS

The present study focuses on a steady-state simulation of a 10° sector of the ARC-class molten salt blanket, a geometry previously used for benchmarking purposes. To clearly separate the physics contributions, the simulations were organized into two main steps:

- Neutronic–CFD pre-calculation:

A coupled neutronic and CFD simulation (without magnetic field) has been first performed to compute the temperature distribution in both the fluid and the solid structures. The neutronic solver provided volumetric power within the blanket geometry. These were used as source terms in the CFD solver, which in turn supplied the thermal–hydraulic fields required for consistent heat transfer modelling. Fixed boundary conditions for the neutronic problem were imposed according to reactor operating conditions; further details on the setup can be found in [8]. The volumetric power deposition field obtained by the neutronic preliminary simulation is shown in Fig. 1.

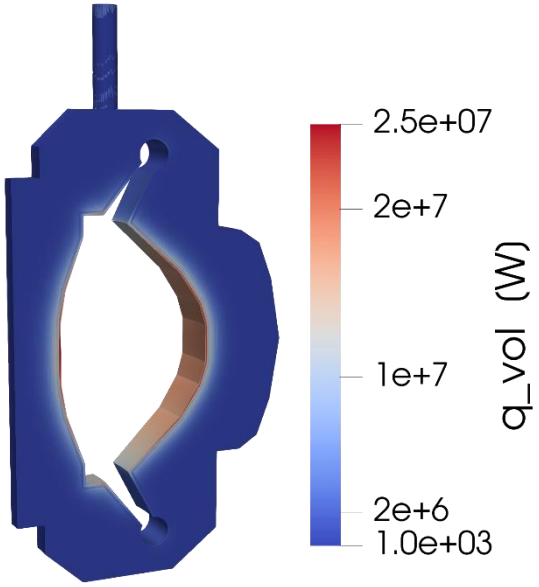


FIG. 1. Volumetric power deposition field obtained by the neutronic preliminary simulation.

Regarding the successive CFD analysis, the coolant (FLiBe) was assumed to enter in the channel through the inlet at 800 K with a uniform velocity of 2.0 m/s. The external surface of the blanket has been treated as adiabatic, neglecting any thermal interaction with the surrounding. The turbulent flow was modelled using the Baglietto-Ninokata k-epsilon model, to improve the near-wall treatment. The main results of this simulation are the flow field (Fig. 2) and the temperature map (Fig.3)

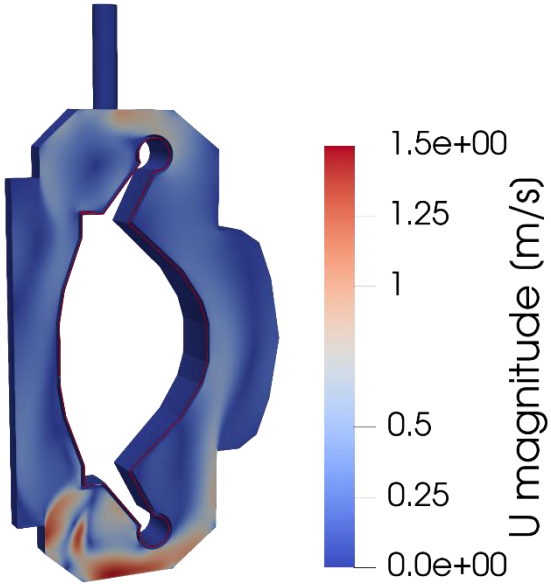


FIG. 2. Flow field of the preliminary simulation without magnetic field. The maximum value of the velocity is limited to 1.5 m/s for the sake of the visualization. The magnitude of the coolant in the channel is one order of magnitude higher.

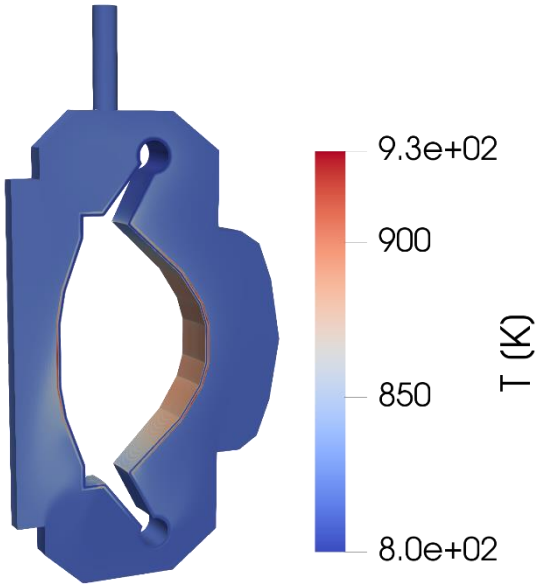


FIG. 3. Temperature map of the preliminary simulation without magnetic field.

- MHD–neutronic simulation:

To reduce computational cost, only the fluid region was simulated, omitting the solid shells. From an electromagnetic perspective, this simplification is justified because the wall conductance ratio c_w of beryllium and Inconel in extremely high. For both beryllium and Inconel c_w reaches tens of thousands, due to their electrical conductivities being several orders of magnitude higher than that of FLiBe and their wall thickness being

comparable to the channel size. Consequently, the walls can be considered effectively perfectly conducting, which allows the solid–fluid interface to fully short-circuit induced currents and correctly represent Lorentz force effects on the fluid. The MHD extension of the turbulence models has been applied to the same 10° sector, with the toroidal 1/R magnetic field profile included. The volumetric heat sources obtained from the neutronic previous calculation were used as input of the simulation. For thermal behaviour, fixed-temperature boundary conditions were imposed at the fluid–solid interface, using the temperature field obtained from the neutronic–CFD simulation. This approximation isolates the influence of MHD effects on the fluid, under the assumption that the thermal field in the solids is not significantly altered by the presence of a finite conducting wall, while still providing physically realistic heat transfer at the interface. Finally, given the non-uniform nature of the magnetic field, and the absence of classical Hartmann- or side-wall boundary layers in the simplified geometry, an MHD boundary layer mesh was generated to ensure sufficient resolution of the electric potential gradients throughout the domain (Fig. 4).

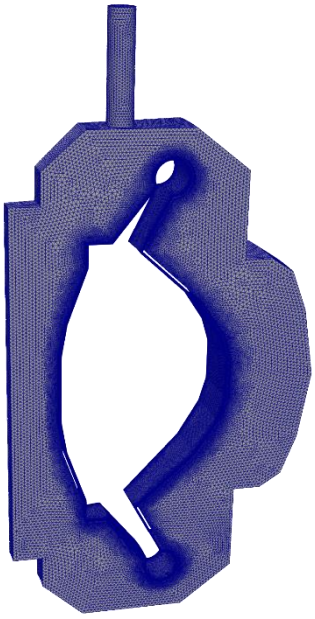


FIG. 4. Schematic of the numerical mesh selected for the simulations.

4. CONCLUSION

Overall, the proposed workflow holds promise for liquid blanket studies in ARC-class fusion reactors. The one-tool approach described here allows a homogeneous treatment of the various physics. The initial results are auspicious: the neutronic calculation produces consistent power fields, the thermal-hydraulics simulation without magnetic field provides reasonable temperature and velocity profiles. The fact that there exists a single, uniform workflow is a significant advantage, as it allows for rapid iteration between simulation stages and comparison of different physical approximations. Having multiple turbulence models enforced also makes it easier to compare different approaches and evaluate how model choice affects blanket behaviour. Full MHD simulations are now underway, and one hopes to be able to report on the status of this work during the poster session. The combined workflow is an important step towards a complete, reliable simulation tool for liquid blankets in fusion reactors and can help designers and researchers in multiphysics analysis of the involved complex systems like the ARC.

ACKNOWLEDGEMENTS

This publication is part of the project PNRR-NGEU which has received funding from the Italian Ministry of University and Research – DM 352/2022.
Eni S.p.A. is gratefully acknowledged for funding the research project and providing computational resources.
Computational resources partly provided by HPC@POLITO (www.hpc.polito.it)
The Massachusetts Institute of Technology is kindly acknowledged for hosting part of the research activities.

REFERENCES

[1] Sorbom, B.N., Ball, J., Palmer, et al., Arc: a compact, high-field, fusion nuclear science facility and demonstration power plant with demountable magnets, Fusion Eng. Des. 100 (2015) 378–405.
[2] Badalassi, V., Sircar, A., Solberg, J.M., Bae, J.W., Borowiec, K., Huang, P., et al., Fermi: fusion energy reactor models integrator, Fusion Sci. Technol. 79 (2023) 345–379.
[3] Fiorina, C., Clifford, I., Aufiero, M., Mikityuk, K., Gen-Foam: a novel OpenFOAM® based multi-physics solver for 2D/3D transient analysis of nuclear reactors, Nucl. Eng. Des. 294 (2015) 24–37.
[4] Harbour, L., Giudicelli, G., Lindsay, A.D, et al., 4.0 MOOSE: enabling massively parallel multiphysics simulation, SoftwareX 31 (2025) 102264.
[5] Baglietto, E., Ninokata, H., A turbulence model study for simulating flow inside tight lattice rod bundles, Nucl. Eng. Des. 235 (2005) 773–784
[6] Baglietto, E., Ninokata, H., 2007. Improved Turbulence Modeling for Performance Evaluation of Novel Fuel Designs. Nuclear Technology 158 (2), 237–248
[7] Meng, Z., Zhang, S., Jia, J., et al., A K-epsilon RANS turbulence model for incompressible MHD flow at high Hartmann number in fusion liquid metal blankets, Int J Energy Res. 42 (2018) 314–320.
[8] Launder, B.E., Reece, G.J., Rodi, W., Progress in the development of a Reynolds-stress turbulence closure, J. Fluid Mech. 68 (1975) 537–566.
[9] Widlund, O., Modeling of magnetohydrodynamic turbulence, PhD thesis, Royal Institute of Technology, Department of Mechanics, 2000.
[10] A. Aimetta, N. Abrate, S. Dulla, A. Froio, Neutronic analysis of the fusion reactor ARC: Monte Carlo simulations with the serpent code, Fusion Sci. Technol. 78 (4) (2022) 275–290
[11] Caravello, M., Aimetta, A., Abrate, N., et al., An OpenFOAM solver for multiphysics modeling of fusion reactor design: the nemoFoam code, Nucl. Mater. Energy 40 (2024) 101693.