Executive summary of the PhD thesis:
“Numerical simulation of MHD flows in breeding blanket and plasma-facing components”

The use of nuclear fusion as an energy source is probably the most complex engineering challenge that humanity is trying to overcome and the enormous benefits that would result from it have led to a constant international commitment to the construction of the first nuclear fusion reactor, sanctioned by the decision to build an international experimental reactor (ITER) in 2006. Since 2014, the European R&D activities in nuclear fusion have been coordinated by the EUROfusion consortium, to achieve the breakthrough goal of building a demonstration fusion power plant (DEMO) after 2050.

Among the huge number of components essential to the reactor operation, two of the key ones are certainly the Breeding Blanket (BB) and the divertor. The first has the task of ensuring the tritium self-sufficiency, the fuel of the reactor, the extraction of the power generated by the nuclear reactions and shielding the other components and personnel from radiation. The divertor has the task of managing and extracting the power and particle exhaust.

Liquid Metals (LMs) are considered attractive solution both as working fluids in blankets, such as for the Water-Cooled Lead Lithium (WCLL) concept, and as “self-healing protection” in advanced divertor and Plasma Facing Components (PFCs). Unfortunately, due to the electrical conductivity of metals, their motion is influenced by the magnetic field used in the reactor to confine the plasma, generating a complex phenomenology which is studied by the liquid metal MagnetoHydroDynamics (LM-MHD) and which must be considered in phase design. These phenomena include the electromagnetic drag, turbulence suppression, modified heat and mass transport and electromagnetic coupling phenomena. In this framework, intense studies and research activities are essential to provide high-quality experimental and numerical data and to develop accurate predictive numerical tools.

The research activity presented in the PhD dissertation “Numerical simulation of MHD flows in breeding blanket and plasma-facing components” aims to contribute to the numerical modelling of MHD phenomena relevant for the BB and for advanced PFCs thought Computational magnetohydrodynamics (CMHD). After an extensive introduction on nuclear fusion technology and LM-MHD, the state of the art of CMHD codes is briefly presented, with a particular focus on the codes used in this research: ANSYS CFX and OpenFOAM.

About the latter, is presented the *phiFoam* solver, a custom tool developed to be able to simulate laminar, incompressible and isothermal MHD flow for ducts with perfectly electrical insulated or perfectly conductive walls, which was validated through both a 2D and 3D benchmark. For the 2D validation, was simulated a square channel with perfectly insulated walls for low and high Hartmann number (Ha), the dimensionless parameter related with the magnetic field intensity, comparing the results with an analytical solution. The code proved capable of predicting both dimensionless mass flow rate and the velocity profile in the MHD layers accurately up to $Ha = 5000$. The 3D benchmark considers a prototypical manifold made from perfectly insulating material and composed of an inlet channel, abrupt expansion and flow distribution in three outlet channels. The flow distribution in the latter was compared with experimental data obtaining errors of the order of 5 % at $Ha = 1503$. Furthermore, the 3D MHD pressure drop caused by the expansion was estimated and compared with that obtained from a semi-empirical model showing an error of the order of 9 %. Overall, phiFoam has been shown to be able to accurately predict the basic phenomena of MHD laminar flows up to high magnetic field intensity.

The *phiFoam* solver has been used to investigate a prototypical manifold, consisting of a feeding pipe, an asymmetrical expansion and three outlet channels positioned at 90° with respect to the expansion, representative of a BB terminal collector. Three different feeding pipe configurations were considered in the range $Ha = 1000 \div 2000$. The flow features are strongly influenced by the generation of axial currents in the expansion, which in turn generate a Lorentz force with both a poloidal and an axial component which significantly impacts the velocity distribution. Furthermore, the presence of the channels attached to the expansion through the right-angled bend in the vertical direction breaks the jet that is created in the upper MHD layer, creating a variety of effects that depend on the point where the jet is broken, and therefore on the feeding pipe position. As this varies, the topology of the vorticity that is created in the expansion and the redistribution of velocity between the outlet channels varies. This significantly impacts the balance of the flow rate between the channels, strongly in favour of the channel aligned with the inlet, not so much for its "privileged" position, as seen for the hydrodynamic case (no magnetic field) where the imbalance is present but much lower with respect the MHD cases, but because of the morphology of the upper jet which carries the bulk of the flow. In any case, the channels distant from the inlet are reached by a flow rate that is too low to guarantee the efficient recirculation of the breeder in the blanket.

Considering the LM spinal manifold, a prototypical co-axial channel representative of the latter has been characterized through numerical calculation by the ANSYS CFX code, varying the intensity of the magnetic field, the geometric parameters and the electrical conductivity of the wall. Then, the electromagnetic coupling phenomenon between the external and internal channel, which are in electrical contact through the conductive wall that separates them, is studied as the intensity of the magnetic field and the distribution of the flow rate between channels vary. Considering perfectly electrical insulated walls, the flow develops in two well-defined cores with the increase of Ha: a fast core in the direction aligned with the magnetic field and a slow core in the direction normal to the latter. These cores match with the wall through the usual MHD Hartmann layers and match each other with an MHD internal layer. With electroconductive wall, the fast core is substituted by two intensive jets close to walls parallel to the magnetic field, with a damped region between these and a more damped slow core compared with the insulated case. By decreasing the electrical conductivity of the walls, the flow features progressively approach those of the insulated case, with the progressive reduction of the jets and the reestablishment of the cores. Increasing the electrical conductivity after a threshold value makes the velocity distribution tends to become uniform and the jets and the cores disappear.

Changing the geometry of the channel by varying the aspect and blockage ratio, the fundamental features of the flow remain unchanged as the magnetic field intensity and the electrical conductivity are constant, but the variation of the gap between the external and internal channel (co-axial channel) determines small variations that intensify when approaching cases in which the gap becomes very small. The co-axial pressure gradient has been correlated with the pressure gradient of an equivalent channel for which exist an analytical solution, developing a correction factor between the configurations. This factor shows an asymptotic behavior for $Ha > 1000$ and allows to estimate the pressure drop for a similar configuration without performing a numerical simulation.

The electromagnetic coupling change considerably the flow features depending on the flow rate repartition between the external and internal channel. The external channel is greatly affected by the electromagnetic coupling phenomenon, which drastically changes the velocity distribution compared to the uncoupled case, already for small values of the internal flow rate. There is the formation of an intense reverse jet and a counter zone flow in correspondence with the side wall shared with the internal channel and a progressive flattening of the velocity profile in the other areas. The internal channel, on the other hand, is much less interested by the coupling, having characteristics close to the uncoupled case even at a very reduced flow rate. By fixing the flow rate in the two channels, as the Hartmann number increases, the typical characteristics of that particular scenario are maintained and all effects are progressively intensified in accordance with the increase in Ha. It is important to note that the counter flow rate under WCLL operating conditions is estimated to be around 28% and must be considered in studies related to the management of the tritium inventory, since fluid recirculation will inevitably lead to tritium accumulation, especially in the outflow manifold. Similarly for the uncoupled case, the pressure gradient for both the external and internal channel has been correlated with the pressure gradient of an equivalent channel for which exist an analytical solution, developing a correction factor between the two configurations. This factor shows and asymptotic behavior for $Ha > 1000$ and allows to estimate the pressure drop for a similar configuration at higher Ha without performing a numerical analysis. These correction factors were used to estimate the pressure drop of the WCLL outboard PbLi spinal manifold that contribute for the 18.5% of the total in-magnet PbLi loop pressure drop.

Considering the PFCs, a thin-film single-phase MHD flow, representative of the armor in a film-type divertor or PFCs, has been investigated with the ANSYS CFX code. The numerical model is validated through a theoretical solution for an insulated chute up to $Ha = 1000$. Consequently, the flow in a chute with insulating, conductive and partially conductive walls has been investigated to highlight the effect of discontinuous wall conductivity on the backing plate and lateral walls. A partially conductive backing plate has a negligible effect on the flow, if also the lateral wall is insulated, consistent to the analogous bounded case, whereas the transition from insulating to conductive lateral walls causes larger pressure losses, higher free surface velocity, structural change in the Hartmann boundary layer and counter flow onset. The location of the conductive sections on the Hartmann wall influences the flow features, resulting in higher free surface velocity and pressure drop when these are close to the backing plate and free surface. The presence of a conductive backing plate with a conductive lateral wall has a great influence on the flow features, greatly enhancing the free surface velocity and pressure drop. These phenomena could be interesting for the PFCs applications, where increasing the free surface velocity with a contained pressure drop could be an attractive solution. In this case, the best compromise is to have a partially conductive lateral wall with the conductive portion placed in the middle/bottom part on the wall, instead of a totally conductive wall.

Finally, is considered the rising of a bubble in a LM. The multi-phase *interIsoFoam* solver present in the OpenFOAM distribution is validated in hydrodynamic conditions for a 2D stationary drop, 2D rising bubble, 3D rising bubble and for the coalescence of two bubbles, compare the results obtained with both numerical and experimental results. The code returned excellent results in almost all the cases tested, but not being particularly accurate for the stationary case, where an error of about 10 % was observed in the evaluation of the pressure jump. This discrepancy is due to an inaccurate resolution of the interface, which generates spurious velocities that impact the force balance between the pressure and the surface tension on the interface. This situation can be mitigated by increasing the number of iterations in the reconstructor-step of the plicRFD method, it is therefore interesting to carry out further simulations considering the improvement of the error with respect to computational time, which increases with the number of iterations.

Then, it was tested for a high-density ratio mixture, simulating the rising of a helium bubble in the LM with different diameters. The solver was able to provide consistent results with the expected regimes in all 5 cases tested. Also in this situation, for low velocities of the surrounding fluid, we can see the impact of the spurious velocities on the interface, which however do not seem to influence the dynamics of the bubble. Overall, the solver has proven to be able to correctly simulate the basic characteristics of a flow with a high-density ratio and therefore it is an excellent candidate for the future implementation of the MHD equations, for the study of the migration of helium bubbles in the blanket but also in the framework of the advanced PFCs.