Thesis Summary

The Dual Fluid Reactor (DFR) is a promising Generation IV reactor concept with numerous advantages. To realize this innovative design, significant efforts are required to achieve optimal design and secure regulatory and public acceptance. While computer codes are commonly used for modeling, actual experiments hold greater credibility than simulations. Therefore, the construction of a mini demonstrator (MD) as an experimental facility targeted to the DFR research is crucial for enhancing trust in computer models dealing with low Prandtl number fluids. By comparing experimental results from the MD with modeling outcomes, validation of the model under various operational conditions provides convincing evidence for regulatory bodies and the scientific community. This validation process enables reliable modeling of the DFR across a wide range of scenarios. In this thesis a preliminary schematic of MD is proposed, and the MD core design is described together with precise dimensions and figures. The case was carefully meshed and exported to calculations based on the relative boundary layers and different scenarios.

The turbulence modeling technique employed in this study showed satisfactory agreement with DNS/LES/Experimental data which increases the confidence in capturing the correct thermal hydraulic behaviors.

Computational results revealed three primary factors influencing heat transfer and fuel velocity in the MD core, two of which are related to system geometry. The positioning of fuel pipe inlets relative to the fuel inlet direction and the placement of coolant pipes in the distribution zone, both significantly affect heat exchange, mass flow rates, and velocity uniformity in the fuel pipes. The third factor was found to be the buoyancy force affecting the direction of the fuel right after entering the MD. The counter flow configuration exhibited more uniform velocity inlets, higher heat exchange efficiency, and uniform temperature gradients compared to parallel flow. In parallel flow, swirling patterns were observed, which, although theoretically enhancing heat transfer, resulted in higher pressure, reduced mass flow rates, and increased risk of vibration-induced pipe fractures and corrosion. The choice of using 3D modelling was crucial as using 2D modelling the resulting wouldn't be able to capture these swirling behaviors, or the observed fuel flow splitting over coolant pipes in the distribution and collection zones.

A major part of the heat transfer was found to occur in the initial contact zone between coolant and fuel in both parallel and counter flow cases. In parallel flow, the heat exchange extends over a larger region, promoting temperature uniformity in the middle core zone. In contrast, counter flow exhibits intense heat transfer in the initial contact zone, which poses safety concerns and mechanical stresses. Mass flow rates differ between parallel and counter flows, affecting heat exchange efficiency, with counter flow configurations found to be more efficient.

The observed discrepancies between parallel and counter flows provide valuable insights for DFR designers. Hot spots were observed over coolant pipes located in front of fuel inlets, which may result in corrosion and mechanical stress issues due to temperature variations. Installation of measuring instruments at specific locations is proposed to capture temperature and velocity profiles, monitor variations in temperature and heat transfer parameters, and gain a better

understanding of fuel and coolant flows within the middle core zone. Certain locations should be avoided for measurement devices, such as areas with backflows or flow separation, to ensure accurate measurements. Finally, the findings and conclusions have shown the importance of modelling the MD that alone gives a better understanding of the DFR thermal hydraulics, and together with the MD as an experimental facility for the purpose of model validation and exploring more different DFR operational scenarios during the reactor design stages.