Modelling of flow and convective heat transfer in serpentine heat exchanger

Kęstutis Račkaitis[a], **Francesco Orlandi**[b], **Robertas Poškas**[a]

[a]*Lithuanian Energy Institute, Nuclear Engineering Laboratory, Breslaujos 3, LT-44403 Kaunas – Lithuania, Email: kestutis.rackaitis@lei.lt*

[b]*University of Modena and Reggio Emilia, Department of Sciences and Methods for Engineering, Via Amendola 2, Pad. Morselli - 42122 Reggio Emilia – Italy*

INTRODUCTION

Condensing heat exchangers play a key role in the waste heat recovery process, which can be of various designs depending on the primary heat source, its pollution level, the place where the heat exchanger needs to be installed, etc. In order to be able to evaluate various influencing parameters, it is most appropriate to use numerical simulation, which can be used to quickly evaluate the efficiency of a heat exchanger of a specific design in the case of different flow regimes. It is the most difficult to model the condensation process, because in this case two phases appear, the interaction of which requires a lot of theoretical and practical knowledge. In order to be able to analyze the interaction of two-phase flows, it is necessary to approach this task step by step, starting from the analysis of single-phase flow to understand the physical processes taking place in the heat exchanger, which usually contains variously arranged tube bundles of different shapes. Therefore, in this paper, numerical studies of hydrodynamics and convective heat transfer were performed to better understand the fluid behavior in a serpentine type condensing heat exchanger. This research study is related to the EU Horizon 2020 iWAYS project (Innovative WAter recoverY Solutions through recycling of heat, materials, and water across multiple sectors) activities.

METHODOLOGY

The presented analysis was conducted in parallel on two different well established CFD softwares. One with Ansys Fluent and the other with Siemens Star-CCM+ CFD softwares. Both simulations considered monophase currents of hot air flowing into the serpentine heat exchanger thermodynamically coupled with the cooling water flowing into the pipes (see Fig. 1). The flows were organized in counterdirections. The metallic pipes were included and characterized by means of thermal conductivity and specific heat of the considered steel material. The cooling water, due to the relatively small increase in temperature, was characterized by a constant density approach. The calculation of the inflowing air Reynolds number led to the intermittent regime to that a turbulent model is required to the calculation. The flowing air than was characterized by realistic thermodynamic properties by means of tabular values. Both the simulations were considered as steady-state. The realized meshes followed two diverse methods. Ansys Fluent case implemented a fully tetrahedral mesh, thus exploiting the ability of this kind of cells to fully adapt to peculiar geometries. Star-CCM+ then implemented both automated and directed meshes methods. The automated method resembles the classic meshing method where the entire volume is meshed by means of macro parameters and user defined refinements or modifications. The directed mesh was fully employed in the regular extrusion of the surface mesh for the cooling water region starting from an automated surface mesh and a patched regular mesh used for the metallic pipes. The hot air region was generated by means of polyhedral cells with the automated method to exploit the same tetrahedral adaptiveness to irregular geometries but reducing the total amount of the employed cells. The Ansys Fluent simulation then resulted in 10 milion cells and Star-CCM+ in a reduced 7 milion cells. The temperature field was recorded by means of numerical probes realized in correspondence of experimental thermocouples, recording a three point average value for the inflowing air, metallic pipes and cooling water.

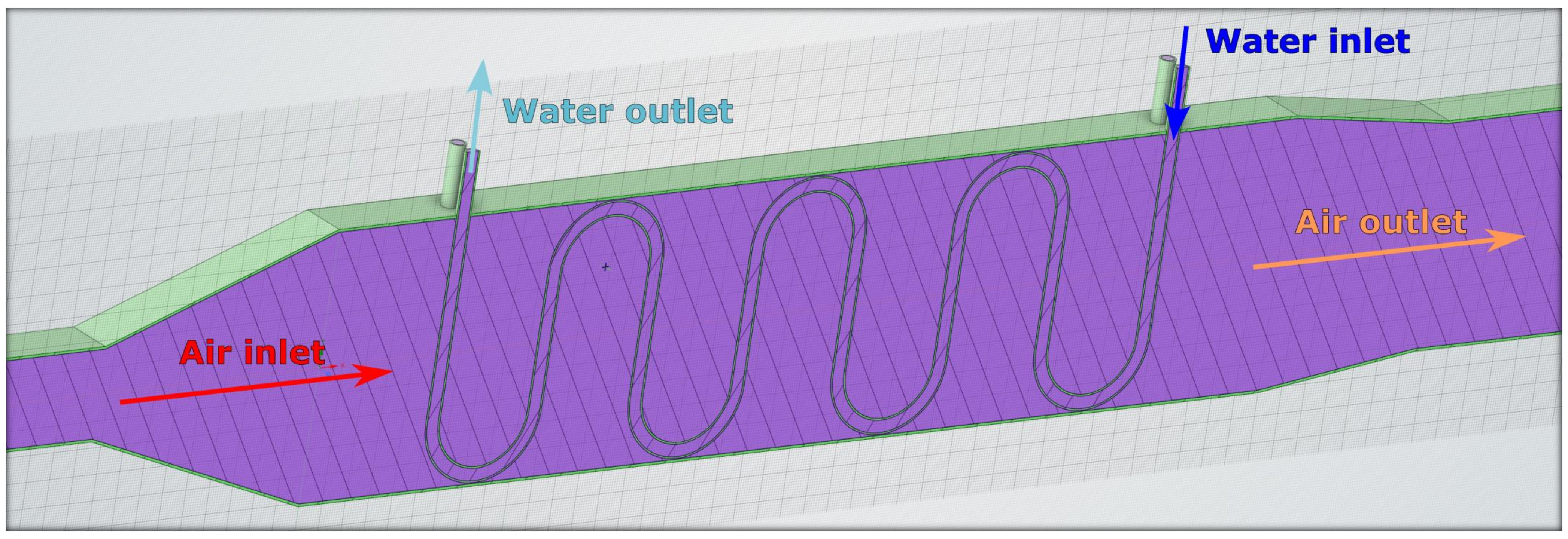

Figure (1): View of the computational domain cut at the vertical plane of symmetry

RESULTS AND DISCUSSION

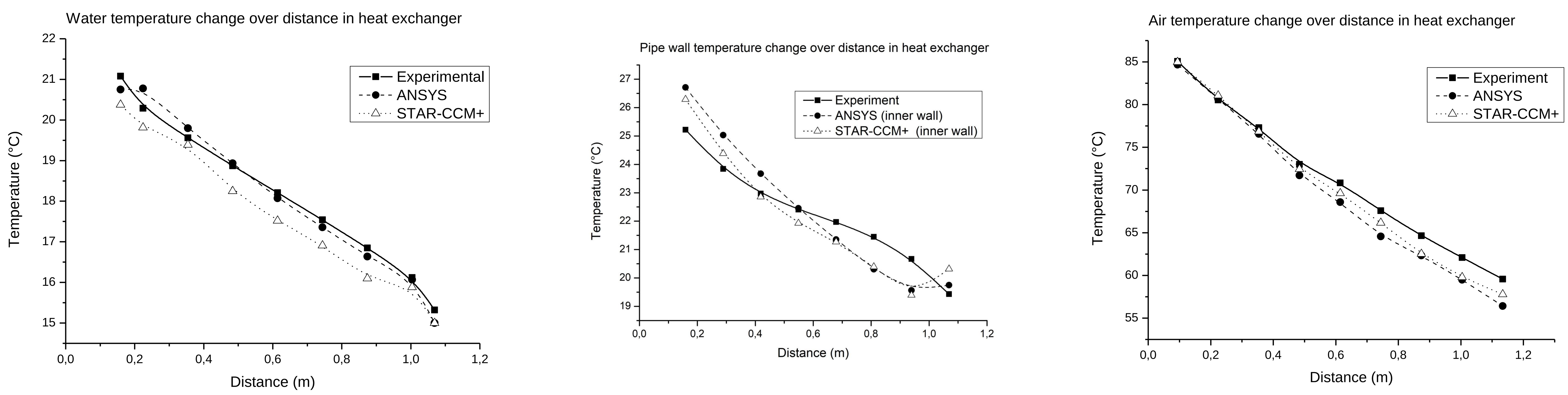

Figure (2): Comparison of calculated results with experimental data: Cooling water temperature change (left); Pipe temperature change (center); Hot air temperature chane (right)

The comparison between the experimental and the simulated results confirmed the good agreement between both the numerical and the experiment (see Fig. 2). Ansys Fluent more precisely predicted cooling water temperature change while Star-CCM+ better predicted air temperature at outlet. A bigger calculation divergence was found on pipe wall where temperature gradient is the biggest. Noted little divergences are perfectly acceptable based on current differences unavoidable between experimental and numerical setups.

LITHUANIAN ENERGY INSTITUTE

Contact information:
Kęstutis Račkaitis
PhD student,
Lithuanian Energy Institute,
Breslaujos st. 3, LT-44403, Kaunas
kestutis.rackaitis@lei.lt
+370 62123721

RAD CONFERENCE
TWELFTH INTERNATIONAL CONFERENCE OF RADIATION, NATURAL SCIENCES, MEDICINE, ENGINEERING, TECHNOLOGY AND ECOLOGY
JUNE 17-21, 2024
HUNGUEST HOTEL SUN RESORT, HERCEG NOVI, MONTENEGRO