



Research Paper

An OpenFOAM-based model for heat-exchanger design in the Cloud

Antonio Gómez^{a,*}, Carlos Montañés^a, Miguel Cámara^a, Ana Cubero^b, Norberto Fueyo^b,
José Manuel Muñoz^c

^a Nablado, S.L., WTCZ Torre Oeste, Planta 11, Avda. María Zambrano 31, 50018 Zaragoza, Spain

^b Fluid Dynamics Group, University of Zaragoza, María de Luna 3, 50018 Zaragoza, Spain

^c Biocurve, S.L., CEEI-Aragón, Nave 8, María de Luna 11, 50018 Zaragoza, Spain



HIGHLIGHTS

- An OpenFOAM CFD model for the simulation of heat exchangers has been developed.
- Its user-friendly interface and high automation enable its use by non-expert users.
- The results of the CFD model have been validated with measurement tests.
- The difference between measured and calculated heat transfer is less than 4.5%.
- The design of an existing heat exchanger has been improved using this CFD model.

ARTICLE INFO

Keywords:

Heat exchanger
Cloud computing
OpenFOAM
User-friendly CFD tool

ABSTRACT

This paper presents a CFD tool, implemented in OpenFOAM and executed in a Cloud environment, to simulate the performance of shell and tube heat exchangers. In spite of the well-known benefits that CFD techniques can bring, these methods are seldom used by small heat exchangers manufacturers as they are unaffordable to them. The high capital costs (licenses and computing resources) and the high level of expertise required hinder a more widespread usage of these techniques. The characteristics of this tool, such as its user-friendly interface, the high level of automation and its implementation in a Cloud environment, allow to overcome the barriers that prevent the access to CFD techniques by most potential users. The CFD model presented in this paper has been validated successfully with measurements in an existing heat exchanger. Further, it has been used to re-design such heat exchanger, achieving a 30% volume reduction, while maintaining its thermal performance.

1. Introduction

Heat exchangers are fundamental parts in many process industries (such as power plants or the chemical and the food industries), and as heat recovery units in the operation of many systems (such as domestic hot water production, space heating or car engines). In most cases, especially in the case of small and medium enterprises (SMEs), the design of heat exchangers relies on the expertise of their technical staff, the construction of prototypes which are improved through trial-and-error cycles, and the use of simple calculation techniques such as the Log Mean Temperature Difference (LMTD) or the Number of Heat Transfer Units (NTU). A suitable design is often found with this methodology, but it is usually far from the optimal one.

The use of Computational Fluid Dynamics (CFD) techniques for the design of heat exchangers has increased in recent years, as reported by Aslam Bhutta et al. [1] in their review of CFD studies of heat

exchangers. Fluid flow maldistribution, fouling, pressure drop and thermal performance are the main areas of analysis with CFD methods. Some examples of these studies are briefly described next. Rosetti et al. [2] studied the effect of the flow maldistribution in the air channel of an open refrigerated display cabinet using CFD. Pal et al. [3] used a CFD model implemented in OpenFOAM to study the heat transfer and flow distribution for shell-and-tube type heat exchangers with and without baffles. Selma et al. [4] built a numerical model using OpenFOAM to optimize the design of a heat pipe exchanger employed in building ventilation systems. An improved design with a 30% lower pressure drop and an increased (by 24%) thermal performance was achieved. Cavazzuti et al. [5] applied a CFD model to optimize a finned concentric pipes heat exchanger, achieving an improvement of the heat transfer capacity of almost 11% without increasing the pressure drop. Lopata and Ocłoń [6] developed a CFD methodology to calculate the fluid flow and heat transfer in fin-and-tube heat exchangers. This model was able

* Corresponding author.

E-mail address: agomez@nablado.com (A. Gómez).

Nomenclature

C_p	heat capacity
F	fouling resistance
G_g	flue gas mass flow rate
h	specific total enthalpy
$\dot{I}_p$	measured thermal power
$\dot{m}$	mass condensation rate
p	pressure
p^S	saturation pressure
T	temperature
$\vec{v}$	velocity vector
X_g^w	water vapor molar fraction in the gas phase
Y_g^w	water vapor mass fraction in the gas phase
Y_g^{wS}	mass fraction of the saturated water vapor in the gas phase

Greek symbols

ρ	density
κ	thermal conductivity
μ	viscosity
α	thermal diffusivity
$(\vec{\tau})$	stress tensor

Δ distance to the interface

Superscripts/subscripts

c	cell center
f	wall interface
g	gas phase
p	a generic phase p
w	liquid (water) phase

Abbreviations

CFD	Computational Fluid Dynamics
P10	minimum load, non-condensing operation mode
P101,P102,P103	experimental results for the operation mode P10
P30	full load, condensing operation mode
P301,P302,P303	experimental results for the operation mode P30
P50	full load, non-condensing operation mode
P501,P502,P503	experimental results for the operation mode P50
SHM	Snappy Hex Mesh
SME	Small and Medium Enterprise
STL	stereolithography file

to determine the effect of fouling on the performance of this type of heat exchangers. Ramos et al. [7] compared the results of a numerical model of a cross flow air-to-water heat-pipe-based heat exchanger with experimental results; the difference between measured data and numerical results was less than 7%. Flaga-Maryanczyk et al. [8] developed a CFD model for simulating a ground source heat exchanger for a passive house ventilation system; the satisfactory agreement between experimental measurements and numerical results led them to conclude that their CFD tool was suitable for the simulation of ground source heat exchangers such as the one proposed in their paper. Wu et al. [9] used a CFD model to research the performance of horizontal-coupled slinky ground source heat exchangers, after its validation with experimental measurements.

Therefore, CFD techniques have been able to simulate a great variety of heat exchangers. Further, carefully-conducted CFD simulations are often in good agreement with the experimental measures, with deviations ranging from 2% to 10% [11].

Moreover, CFD techniques allow an important reduction of the time and costs required to design heat exchangers, since the performance of alternative designs can be analyzed through the simulation of virtual heat exchangers without the need for the construction of prototypes. The accuracy and cost-effectiveness of CFD techniques make them very useful design tools.

In spite of these advantages, there are two considerable barriers to the use of CFD tools by an SME: the high capital costs (software licenses, large computing resources) and the expertise required to use these tools. The uncertain added value of these tools in the design process often makes SMEs reluctant to invest in such tools.

The primary objective of the work presented in this paper has been the development of a CFD tool to simulate shell and tube heat exchangers that may enable SMEs to use these calculation methods. To build this tool, a numerical model, based on open source CFD software, has been developed and, subsequently, it has been integrated into a Cloud application. The result is a user-friendly and highly automated CFD tool that is executed in the Cloud. An additionally aim of this work is to demonstrate the benefits that can be brought about by this kind of tools. In order to do this, the CFD tool has been applied to an existing heat exchanger, manufactured by an SME.

This CFD tool is an innovative contribution that shows the potential

of these approaches to expand the use of Computational Fluid Dynamics. It consists of a complex multi-domain model (the gas and water sides of the heat exchanger are solved simultaneously; Pal et al. [3] and Leoni [10] considered only one domain to avoid this complexity), with a water vapor condensation model that has been implemented in the gas-side domain; the tool is fully automated and integrated into a Cloud application. In order to show the reliability and the usefulness of this tool, we apply it to a real industrial case: the condensing heat exchanger of a biomass boiler manufactured by a SME. In this real experience, the numerical results obtained are compared to experimental measurements, and improvements to the current heat exchanger design are tested and analyzed. Further, the performance analysis and the potential improvements featured in this paper can be instrumental in the development of new designs for this type of heat exchanger.

The structure of this paper is as follows. The next section, materials and methods, describes the condensing heat exchanger selected for testing the CFD tool, the experimental set-up used to obtain empirical data, the numerical model developed and the cloud application. Then, the validation work is discussed and, subsequently, the design improvements of the condensing heat exchanger, achieved with the CFD tool, are presented. The paper ends with some conclusions.

2. Materials and methods

2.1. Heat exchanger description

The CFD tool presented in the paper has been applied to the heat-exchanger of a commercial 25 kW_{th} condensing biomass boiler. These boilers have higher efficiencies than standard ones since they use the latent energy in water vapor through its condensation. Until recently, condensing boilers have been available only for gas and oil firing; biomass boilers are now starting to be fitted with this technology.

The heat exchanger employed to test the CFD tool is one of the key components of the condensing boiler. This patented heat exchanger is the first to be arranged in a biomass boiler as a cluster of curved pipes, instead of straight ones. This layout provides a much larger heat transfer surface area (up to 50% over a standard exchanger), and a turbulent flow without additional aids such as turbulators. Both the

Download English Version:

<https://daneshyari.com/en/article/7045121>

Download Persian Version:

<https://daneshyari.com/article/7045121>

[Daneshyari.com](https://daneshyari.com)