Available online at www.sciencedirect.com

ScienceDirect

journal homepage: www.elsevier.com/locate/he

CFD modeling of a diesel evaporator used in fuel cell systems

M. Atmaca [a,*], İ. Girgin [b], C. Ezgi [b]

[a] Department of Mechanical Engineering, Marmara University, Technology Faculty, Turkey
[b] Department of Mechanical Engineering, Turkish Naval Academy, Istanbul, Turkey

ARTICLE INFO

Article history:
Received 26 November 2015
Received in revised form
22 February 2016
Accepted 23 February 2016
Available online 17 March 2016

Keywords:
Diesel
Nitrogen
Evaporator
Heat transfer
CFD
Modeling

ABSTRACT

Diesel evaporators are one of the key components of diesel fuel processing for fuel cells. Diesel evaporator is required to evaporate diesel fuel in diesel fueled fuel cell systems. In this study, diesel evaporator which is a spiral-tube heat exchanger via indirect heat supplied by hot nitrogen gas is modeled and the fractional distillation curve results of diesel fuel are compared to experimental data reported in literature. In this work, the ANSYS Fluent 14.0 Computational Fluid Dynamics (CFD) code is used to simulate the 3-dimensional, turbulent, two-phase, multi-component and reacting flow-field, developed in a diesel evaporator.

Introduction

Fuel Cell Systems are clean energy systems. Hydrogen is an ideal fuel for fuel cells because of its high reactivity and zero emission characteristics. However, it may not be feasible to store the hydrogen in some vehicles because of the higher volume requirements, high fuel storage cost and low energy storage efficiency and security problems.

Hydrogen can be produced from any hydrocarbon fuel, various biological materials and from water. Hydrocarbon fuels, such as Liquefied Petroleum Gas (LPG), gasoline and diesel, are widely being studied for hydrogen production due to the already available infrastructure of these fuels.

Diesel is a common fuel source for transportation application worldwide. Diesel is one of the best hydrogen storage systems, because of its very high volumetric hydrogen density and gravimetric density. This makes diesel reforming an attractive option for hydrogen production on-board surface ships such as land based vehicles.

The first stage of a compact fuel processing system is the evaporation of diesel fuel before it enters to the hydrodesulphurization unit. A significant portion of the energy required for the whole process may be consumed in this step in the case of liquid fuel with high boiling point interval.

Diesel is a complex mixture of hundreds of hydrocarbon compounds like paraffins, olefins, and aromatics. There are three methods for evaporating diesel fuels. The first approach is liquid fuel atomization, in the form of a spray, is commonly used in a large variety of technical combustion applications. Second approach is Stabilized Cool Flame regime which has been proposed by Lucka and Koehne [1]. In this case, two main

* *Corresponding author*. Tel.: +90 2163365770, +90 5336967561 (mobile); fax: +90 2163378987.
E-mail address: matmaca@marmara.edu.tr (M. Atmaca).
http://dx.doi.org/10.1016/j.ijhydene.2016.02.122
0360-3199/

phenomena, namely droplet evaporation and fuel combustion, are essentially separated; the liquid fuel is allowed to evaporate (but not to burn) in an open-flowing chamber where low temperature oxidative cool flame reactions occur. Kolaitis and Founti modeled a diesel oil evaporation device operating in the "stabilized cool flame" regime [2]. Third approach is to evaporate diesel fuel in an evaporator by electrical energy or hot gas [3].

Kristyadi [4] modeled the heating and evaporation of fuel droplets. Evaporation of droplets involves simultaneous heat and mass transfer processes. The heat required for evaporation is transferred to the droplet surface by convection and radiation from surrounding hot gases while fuel vapor is transferred by convection and diffusion back into gas stream. Lefebvre [5] has considered steady state and unsteady state evaporation.

Power systems for the navies of NATO countries operate on naval distillate fuel (NATO symbol F-76). This fuel is characterized as 385 °C (max.) end boiling point and 12.5% wt. (min.) hydrogen content diesel fuel.

Krummrich et al. [6] reported that the DESIRE-project was performed from 2001 to 2004 to demonstrate the feasibility of the reforming of F76 diesel fuel into a hydrogen-rich gas suitable to be utilized in Proton Exchange Membrane (PEM) fuel cells. They performed several studies on different evaporator designs because of the risk of formation of carbonaceous material.

Sarioglan et al. [3] stated that there was no commercial diesel evaporator in the market for fuel processors integrated with low temperature fuel cells. Hitherto, there has been no commercial diesel evaporator in the market. They evaporated F76 diesel fuel in a tube-and-tube heat exchanger which consists of two concentric stainless steel spiral tubes via indirect heat supplied by hot nitrogen gas. They obtained the fractional distillation curves of original and product F-76 diesels produced in the evaporator. In that evaporator, energy needed for evaporation was provided from a hot nitrogen gas stream and evaporation was carried out at 390 °C and 4 bar (absolute) with 17 and 98 ml/min flow rates. The use of hot nitrogen simulates the operating conditions of the targeted fuel processing system in which hot waste gases can be used for the evaporation of the liquid fuels. In real applications, energy needed for the evaporation of diesel can be provided from diesel engine exhaust gases. Hot nitrogen has been used in this numerical study since it also has been used in the experimental work by Sarioglan et al. [3] to compare the results of both works.

Ezgi et al. [7] designed and thermodynamically analyzed a 120 kW F-76 diesel-fueled solid oxide fuel cell system (SOFC) as an auxiliary engine on-board a naval surface ship.

Helical and spiral coils are curved pipes which are used as curved tube heat exchangers in various applications such as dairy and food processing, refrigeration and air conditioning industries, and heat recovery systems. The heat transfer coefficient is higher in a spiral tube than in a straight tube. Spiral-tube heat exchangers are suitable for thermal expansion and clean fluids, since cleaning is almost impossible [8].

Due to advances in computer hardware and software and consequent increase in calculation speed, the Computational Fluid Dynamics (CFD) modeling technique was developed as a powerful and effective tool for better understanding of the complex hydrodynamics in many industrial processes.

Bredell et al. [9] investigated the performance of fan in a forced draft air-cooled steam condenser. A CFD model was developed by Erek et al. [10] to verify the influences of fin geometry changes on heat transfer and pressure drop behavior of plate fin and tube heat exchanger. Zhang et al. [11] examined the heat transfer characteristics of a helically baffled heat exchanger combined with a finned tube experimentally and using CFD modeling.

Eiamsa-ard et al. [12] have carried out a numerical study on a tube equipped with loose-fit twisted tapes. They used four turbulence models and their results showed that prediction results of the Shear Stress Transport (SST) $k-\omega$ turbulence model have better agreement with measurement results compared with other models. The increase of heat transfer rate caused by metallic porous tube inserts was investigated by Pavel and Mohamad [13].

The aim of this study is to model an evaporator to be used as a part of a diesel fuel processor using Computational Fluid Dynamics (CFD) software ANSYS and the fractional distillation curve results of diesel fuel are compared to experimental data reported in Sarioglan et al. [3].

CFD modeling

In this investigation, a three-dimensional numerical simulation of the heat transfer is conducted using the CFD code ANSYS Fluent 14. The modeling is carried out in order to predict and explain the experimental observations.

The continuity, momentum and energy equations together with scalar transport equations are used to obtain fluid flow pattern and heat transfer rate. To account for the effects of smaller scales of motion, the RNG model was developed using Re-Normalisation Group (RNG) methods by Yakhot et al. [14] to renormalise the Navier–Stokes equations. In the present study, the RNG version of $\kappa-\varepsilon$ turbulence model [15] with enhanced wall functions for the near wall treatment is used to model the turbulent flow regime.

To simplify numerical simulation while still keeping the basic characteristics of the process, following assumptions are made: (1) the fluid flow and heat transfer processes are turbulent and in steady-state; (2) convection heat transfer coefficient is supposed constant for fuel and nitrogen (3) the natural convection induced by the fluid density variation is neglected; (4) the evaporator is well-insulated hence the heat loss to the environment is totally neglected [16,17].

Table 1 – Specifications of the case evaporator.

Type	Tube and tube
Shell side fluid	Diesel fuel
Tube side fluid	Hot nitrogen
Outer tube diameter (D)	10 mm
Inner tube diameter (Di)	6 mm
Spiral tube length (L)	6 m
Outer cylindrical tube height (h_t)	600 mm
Outer cylindrical tube diameter (D_t)	150 mm
Arrangement	Counter flow
Tube material	316-stainless steel