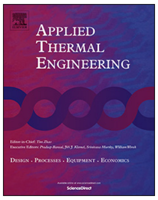
Applied Thermal Engineering 105 (2016) 979–987

Contents lists available at ScienceDirect

Applied Thermal Engineering

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



Research Paper

CFD analysis of drift eliminators using RANS and LES turbulent models

J. Ruiz [a,*], A.S. Kaiser [b], B. Zamora [b], C.G. Cutillas [a], M. Lucas [a]

[a] *Departamento de Ingeniería Mecánica y Energía, Universidad Miguel Hernández, Avda. de la Universidad, s/n, 03202 Elche, Spain*
[b] *Departamento de Ingeniería Térmica y de Fluidos, Universidad Politécnica de Cartagena (Campus Muralla del Mar), Dr. Fleming, s/n, 30202 Cartagena, Spain*

HIGHLIGHTS

- The computational expense is 7–10 times higher for LES simulations.
- RANS models enhanced with a DPT model predict successfully the eliminator performance.
- Small differences between LES and RANS have been observed for the eliminator simulations.

ARTICLE INFO

Article history:
Received 27 November 2015
Accepted 25 January 2016
Available online 13 February 2016

Keywords:
Cooling tower drift
Drift eliminator
CFD
RANS
LES

ABSTRACT

Drift eliminators design should guarantee high collection efficiencies, for preventing cooling tower emissions, and low induced pressure losses, to reduce the energy consumption of the cooling system. CFD methods have become the main method to design drift eliminators. One of the major issues when predicting drift eliminator collection efficiency is to model the interaction between the water drops and the turbulent eddies. This paper aims to determine the best numerical approach to predict the performance of drift eliminators. The Reynolds Average Navier–Stokes (RANS) and Large-Eddy Simulation (LES) turbulence approaches are considered. Calculations are performed for a lath-type drift eliminator considering three different aspect ratios and a wide range of velocities and droplet diameters. The numerical results have been validated through experimental data. The computational expense is much higher for the LES approach (7–10 times). The RANS approach enhanced with a turbulent dispersion of droplets model has proven successful to appropriately predict the performance of the eliminator predicting almost the same results as LES but cost effective. No substantial differences are found between the predicted results by LES and RANS approaches: less than 3% for the pressure drop and 7% for the collection efficiency on average.

1. Introduction

Cooling tower drift emissions are harmful for several reasons but mainly because they may affect the human health and the environment. For that reason and in order to minimize the amount of water escaping the tower in the exhaust air stream, elements known as drift eliminators are set at the exit surface of cooling towers. Drift eliminators work by changing the direction of the airflow as it passes through them and collecting water droplets by inertial impact. Therefore, the eliminators performance can be quantified mainly by the droplet collection efficiency and the pressure drop induced in the airstream. High-pressure losses contribute to lower airflow rates in natural draft cooling towers (buoyancy driven flow) or higher engine power consumption of the fans in mechanical draft cooling towers. These facts lead to obtain lower overall efficiency values for the condensed system. In this sense, the best eliminator design must pool these two effects.

With respect to the eliminators design, Computational Fluid Dynamics (CFD) techniques have become the main method to perform this task. CFD is based on solving the governing equations that describe the spatial and temporal evolution of flows. Every turbulent flow exhibits an irregular behavior both in space and time because it contains spatial (coherent) structures that develop in time. These structures are often referred to as eddies, as they are usually associated with rotating motions of fluid. One fundamental result of turbulence theory is that these eddies are not all of one particular size, but that a (broad) continuous range of large to small eddies exist in every turbulent flow. Theoretically, all the turbulent flows can be solved by resolving all the scales present in the fluid. In this sense, the mesh has to be fine enough to resolve the smallest length

---

* Corresponding author. Tel.: +34 965222433; fax: +34 965222493.
E-mail address: j.ruiz@umh.es (J. Ruiz).

http://dx.doi.org/10.1016/j.applthermaleng.2016.01.108
1359-4311/

and time scales (Kolmogorov scales), respectively. This is known as Direct Numerical Simulation (DNS). Nieuwstadt et al. [1] calculated the required computer memory and the computer time in order to perform a realistic DNS calculation. However, even with the capacities of today's supercomputers, DNS is only feasible for simple geometries and low Reynolds numbers. A second approach to solve turbulent flows by numerical means is the well known Reynolds Averaged Navier–Stokes (RANS). The time-averaging of the Navier–Stokes equations is employed to reduce the range of scales present in the flow. Since time-averaging is larger than the largest scale of turbulent fluctuations, the equations of motion describe the evolution of the mean flow. The influence of the removed scales is incorporated into the so-called Reynolds stress tensor. RANS modeling allows us to solve a wide range of engineering problems due to the reduced number of operations required for achieving a solution. The Large-Eddy Simulation (LES) turbulent approach arises as a remedy to overcome DNS and RANS limitations. In LES, the large eddies are solved, whereas the small eddies are removed by a spatial filtering procedure (usually by mesh size). This is justified because the large eddies are more dependent than the flow configuration. Thus, only the effect of the removed small scales remains to be modeled. This modeling is usually referred to as SubGrid-Scale (SGS).

Concerning drift eliminator analysis by numerical simulation, Chan and Golay [2] developed a numerical model to investigate the collection efficiency and the pressure drop for different types of drift eliminator. They assumed laminar flow and the free-slip and no-slip conditions were imposed on the walls. They suggested a selection (design) criterion that consisted of setting a pressure drop limit across the eliminator and choose the geometry yielding to the best collection efficiency. Verlaan [3] used the PHOENICS code to predict the flow and collection efficiency on a vane-type drift eliminator. He used a low Reynolds number k–ϵ turbulence model. He optimized the geometry of the eliminator and reduced the pressure drop by 50% without loss of collection efficiency. Wang and James [4] reported a numerical model for the collection efficiency of two wave-plate demisters. They obtained a fair agreement between the predicted and the experimental results using a low Reynolds number k–ϵ turbulence model, although large discrepancies were found over a range of droplet sizes. They justified these discrepancies due to the turbulent motion. Sriveerakul et al. [5] studied three types of drift eliminators in terms of the dimensionless pressure drop and the collection efficiency using the FLUENT code. They suggested the use of the inertial parameter rather than the droplet diameter when comparing eliminators in terms of separation efficiency. Zhao et al. [6] and Narimani and Shahhoseini [7] obtained models for predicting the eliminator performance based on response surface methodology. They mainly investigated and related the separation efficiency with structural parameters of the vane. Rahimi and Abbaspour [8] and Kouhikamali et al. [9] developed numerical models to predict the performance of wire mesh drift eliminators.

The abovementioned studies did not include the turbulent dispersion of droplets exerted by the mean flow. Wang and James [10] used the CFX code to simulate the continuous phase, whereas they developed an algorithm to simulate the Lagrangian particle tracking. These authors found that a refinement of the Eddy Interaction Model (EIM), called "varied EIM" and based on the suggestions of Kallio and Reeks [11] and Sommerfeld et al. [12], yielded results close to experimental data employing the k–ϵ turbulence model. Galletti et al. [13] investigated the performance of two types of commercial wave-plate drift eliminators with zig-zag profile. They compared the predicted results by a low Reynolds number k–ϵ and the SST k–ω turbulent models. They used the "varied EIM" turbulent dispersion model. Zamora and Kaiser [14] presented a systematic study of drift eliminators performance. They compared four types of drift eliminators by numerical means using the Shear–Stress Transport (SST) k–ω turbulence model enhanced with an EIM. They proposed a global correlation for the collection efficiency as a function of the inertial parameter and the removal geometric parameter, introduced in their work.

All the numerical studies cited above used the time-averaging of the Navier–Stokes equations when carrying out the numerical simulations. Even though some of them modeled the interaction between water droplets and the turbulent scales using the RANS approach, this factor is critical in order to predict accurately the collection efficiency of the eliminator according to the literature review. By solving the relevant turbulent scales of motion, the LES approach can be a powerful method to predict correctly the droplet deposition in drift eliminators. Eggels [15] used the LES (and also the DNS) approach to simulate both, standard and rotating pipe flows, in order to investigate the statistical properties of such flows and to validate the conventional Reynolds stress turbulence models. Pittard [16] developed a LES based, numerical model of turbulent pipe flow with pipe structural analysis purposes. With regard to particle deposition problems, Breuer et al. [17] and Berrouk and Laurence [18] developed numerical models to predict particle deposition in 90° circular cross-section bends.

So far, the literature review has highlighted how relevant is to model appropriately the interaction between the water droplets and the turbulent eddies when predicting the drift eliminator collection efficiency. However, no studies regarding LES simulations in drift eliminators have been found in the reviewed literature. Therefore, the main objective of this paper is to determine the best numerical approach to predict the performance of drift eliminators. For this purpose, the results predicted by CFD using RANS and LES turbulent approaches are compared when predicting the wall-bounded shear driven flow inside a drift eliminator. The performance of a lath-type eliminator is evaluated by calculating the collection efficiency and the dimensionless pressure drop coefficient. Three different aspect ratios ($b/L = 1/3$, $1/4.5$ and $1/6$) and a wide range of velocities and droplet diameters are considered in the simulations.

2. Methodology

2.1. Experimental test facility

In order to achieve an appropriate validation of the stated numerical model, a wind tunnel experimental facility was designed and built ad hoc to perform pressure drop experimental tests. The facility consists of three elements: the test section, the nozzle and the diffuser. The test section has a cross-sectional area of 0.49×0.7 m^2 and a length of 3.5 m. The nozzle has a cross-sectional area of 1.2×1.7 m^2, a length of 1.55 m and an area ratio equal to 6. The diffuser area ratio is equal to 2.3. A full description of the experimental test facility can be found in Ruiz [19].

A lath-type L-shaped drift eliminator was studied in the experimental facility. It consists of a set of fiberglass plates, with a zigzag structure in which the airflow direction is modified 3 times (180°), separated by 0.025 m. The total length of this eliminator is 0.15 m (aspect ratio equal to 1/6) and the lath thickness is 0.003 m. This eliminator has been referred to as drift eliminator C in the works of Lucas et al. [20,21].

2.2. Physical model

2.2.1. Physical domains

Two different domains are used in the simulations. With respect to the experimental validation, a 2.5 m long, 0.5 m wide and 0.7 m hight domain is considered (Fig. 1). It reproduces a portion of the experimental facility where the tests were performed. To compare RANS and LES turbulent approaches, however, a single channel domain is employed (Fig. 2). The LES simulations for the wind tunnel