Applied Thermal Engineering 127 (2017) 157–164

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

A numerical investigation of vapor flow in large air-cooled condensers

Michael Owen *, Detlev G. Kröger

Department of Mechanical and Mechatronic Engineering, University of Stellenbosch, South Africa

HIGHLIGHTS

- A novel ACC vapor flow numerical simulation method is presented.
- Tube inlet loss coefficient distributions determine the flow distribution in the ACC.
- The subsequent demands on dephlegmator performance are likely higher than expected.

ARTICLE INFO

Article history:
Received 27 February 2017
Revised 17 July 2017
Accepted 5 August 2017
Available online 7 August 2017

Keywords:
Air-cooled condenser
Vapor flow distribution
CFD
Modelling
Simulation
Inlet losses

ABSTRACT

A numerical simulation method – using a combination of computational fluid dynamics (CFD), numerical and analytical methods – for investigating the nature of the vapor flow distribution in the primary condensers of a large ACC is presented. A definite trend in inlet loss coefficient distribution through the heat exchanger bundles is identified. The vapor flow distribution is found to be influenced by the inlet loss coefficients to such an extent that the flow distribution appears to not conform to the expected pattern typical of parallel or reverse flow manifolds. This non-uniform axial distribution of vapor amongst the primary condenser tubes places an additional demand on the dephlegmator that cannot be overlooked. This verified simulation method can be applied during the design phase of an ACC to improve the steam-side performance of these systems.

1. Introduction

1.1. Background and motivation

Air-cooled steam condensers (ACCs) are a mature technology finding increasing application in the power producing industry. A growing interest in using ACCs in low temperature cycles (solar thermal, geothermal and organic Rankine cycles), and the ever increasing need to reduce energy production related water consumption means that there is continuing scope for improvement in the design and performance of these systems.

The vast majority of recent research relating specifically to ACC performance focuses on the air-side operation and the effects of ambient and meteorological conditions in particular (see for example [1–3]). Few studies have addressed steam-side performance considerations specific to ACCs and information regarding the vapor flow distribution through ACC heat exchanger bundles could not be found in the literature. CFD studies of the steam side are generally not carried out due to the high complexity and computational expense. Dynamic process simulations such as [4] are often carried out on the vapor cycle but these require the loss coefficients to be specified as inputs. Understanding the nature of the vapor flow through an ACC heat exchanger is important for addressing the issue of non-condensable gas accumulation through appropriate dephlegmator (deaerator) sizing, thus ensuring reliable and predictable ACC performance, and in light of the desire to achieve lower steam-side pressure drops resulting in enhanced cooling performance and subsequently greater turbine output. Accurate loss coefficient data is also necessary to improve dynamic process simulation accuracy.

1.2. A review literature relating to ACC steam-side operation

The important topics of in-tube condensation heat transfer and pressure drop are addressed by [5–9] amongst several others. Heat transfer in the presence of non-condensable gases – see for example [10–12] – and flooding in reflux condensers – see [13–15] – are also well researched.

Flow distributions in heat exchanger-type manifold systems have been addressed in several texts including [16], and more

* Corresponding author at: Department of Mechanical and Mechatronic Engineering, Stellenbosch University, Private Bag X1, Matieland 7602, South Africa.

E-mail address: mikeowen@sun.ac.za (M. Owen).

http://dx.doi.org/10.1016/j.applthermaleng.2017.08.026
1359-4311/

Nomenclature

Symbols

a	header dimension (m)
b	header dimension (m)
C	coefficient
d	diameter (m)
f	friction factor
K	loss coefficient
L	length (m)
p	pressure (N/m^2)
v	velocity (m/s)

Greek symbols

ρ	density (kg/m^3)

Subscripts

d	discharge
ds	downstream
eq	equivalent
H	hydraulic
i	inlet
us	upstream

recently [17]. Berg and Berg [18] investigated vapor backflow due to row effects in multi-row, single vapor-pass, air-cooled condensers with isothermal vapor flow. Breber et al. [19] used Berg and Berg's method to develop equations describing the effect of backflow on the cooling performance of air-cooled heat exchangers. Fabbri [20] modified Berg and Berg's model to account for the influence of changing vapor temperature and non-uniformities in the air-flow distribution over the heat exchanger.

Unfortunately [18], and subsequently [19,20], ignore inlet/outlet and acceleration effects caused by condensation inside the tubes in their models. In addition, very few of the above mentioned studies are based on parameters and conditions similar to those characteristic of large scale ACCs for power plant applications.

1.3. Purpose of this study

This study aims to create a numerical simulation method – using a combination of computational fluid dynamics (CFD), numerical and analytical methods – for investigating the nature of the vapor flow distribution in the primary condensers of a large ACC. The simulation method can be applied to existing ACC systems to analyse the nature of losses and the accumulation of non-condensable gases and in so doing can assist in identifying the causes of steam side problems that lead to reduced ACC performance. For design purposes, the model can contribute towards reducing steam-side losses and proper dephlegmator sizing resulting in improved and more reliable performance. No other study of this nature could be found in the literature.

2. System description

A practical hypothetical ACC for a direct-cooled power plant is considered as a case study. The ACC consists of several "streets" of six A-frame condenser units or cells, as shown in Fig. 1(a). Steam is fed via a distributing manifold to the dividing header of the so-called primary condenser units (units 1, 2, 4, 5 and 6) which are connected in parallel between common headers as illustrated in Fig. 1(b). Partial condensation takes place in a co-current vapor/-condensate flow arrangement in these primary units. Excess vapor leaving the primary units is condensed in the secondary reflux (counter-current vapor/condensate flow) condenser, or dephlegmator (unit 3). The dephlegmator is connected in series with the primary condenser units (Fig. 1(b)) and must provide sufficient suction to prevent vapor backflow and the subsequent accumulation of non-condensable gases in the primary condensers. Units 1 and 2 have a parallel (Z-type) manifold configuration while units 4, 5 and 6 have reverse (U-type) manifold configurations.

Each ACC unit consists of 8 finned-tube two-row heat exchanger bundles (Fig. 2). Table 1 gives the tube dimensions and heat exchanger parameters. The bundles are arranged in an A-frame configuration with a half-apex angle of θ = 30° above an axial flow fan (d_F = 9.15 m).

3. Modelling

A single CFD model of an entire ACC street is impractical due to the size and complexity of the flow domain. In this case for example, a single primary condenser unit contains 920 tubes with a length to hydraulic diameter ratio of $L/d_h \approx 320$. The size of the mesh required for accurate resolution of such a unit, let alone multiple coupled units, is prohibitive.

A modelling procedure was therefore developed which consists of:

- CFD models of the heat exchanger dividing and collecting headers belonging to each of the 5 primary condenser units.
- An analytical representation of the tubes connecting the dividing and combining headers.
- A numerical flow distribution calculation code that combines the CFD and analytical models to simulate the flow distribution in the primary condensers.

The numerical solution implements an iterative solution procedure as follows:

(a) Tube inlet loss coefficients in the heat exchanger bundles are determined from dividing header CFD data generated assuming a uniform vapor flow distribution amongst the tubes. For the first iteration the flow rate entering each tube is taken as being equal to the condensation rate for the relevant tube row (which is in turn determined from the thermal performance characteristics of the tubes).
(b) The inlet loss coefficients are implemented in the numerical flow distribution calculation code and a new heat exchanger tube flow distribution is determined.
(c) The updated flow distribution is applied in the CFD model and new inlet and outlet loss coefficients (from the combining header models) are predicted.
(d) Steps (b) and (c) are repeated until the flow distributions predicted in subsequent iterations converge.

3.1. CFD model

3.1.1. Description

Each primary condenser unit was modelled separately, starting with the upstream unit (1) and moving to the downstream unit (6) in a sequential manner.