Experimental and numerical investigation of optimum design of semi industrial heat recovery steam generator inlet duct

Pedram Hanafizadeh a, Mojtaba Mirzakhani Siahkalroudia, Pouria Ahmadi b,⇑

a Center of Excellence in Design and Optimization of Energy Systems, School of Mechanical Engineering, College of Engineering, University of Tehran, Tehran, Iran
b Fuel Cell Research Lab (FCReL), Mechatronic System Engineering, Simon Fraser University (SFU), Vancouver, Canada

HIGHLIGHTS

• To suggest geometry improvement of inlet duct of heat recovery steam generators in power plants.
• To gain optimum distribution of inlet duct of pilot model.
• To use both numerical and experimental methods for inlet duct geometry optimization.
• To apply the proposed method in a standard inlet duct of a 5 MW HRSG in power plant.

ARTICLE INFO

Article history:
Received 8 December 2015
Revised 9 April 2016
Accepted 4 May 2016
Available online 5 May 2016

Keywords:
Heat Recovery Steam Generator (HRSG)
Optimum inlet duct
Uniformity

ABSTRACT

The optimization of Heat Recovery Steam Generators (HRSGs) is one of the key parameters to optimize the efficiency of combined cycle power plants. One of the major characteristics in the field of HRSG design and optimization is the investigation of flow within the body from the hydrodynamics point of view. Making uniform flow with minimal pressure loss over time has been an interesting issue for HRSG designers and manufacturers. In this research study, a comprehensive analysis and assessment of flow patterns in the tradition zone of inlet duct of horizontal HRSG is conducted using both experimental and numerical methods. The HRSG is modeled in 1/10 scale-down and velocities are measured by an anemometer device (KM 909 model) at diverse points in its selected sections. Numerical simulation is performed to compare with different results of experimental study in order to ensure the validity. In this study, the difference rages of inlet duct angle are analyzed and finally the optimum model in terms of flow was compared with initial model by using smoke test. The results shown that inlet duct with 23° angle and movable plate with 200 mm length is better than others based on hydrodynamics with minimum velocity deviation and pressure loss between the outlet of fan and transition zone.

1. Introduction

Combined cycle power plants (CCPPs) have attracted ample interests during the last decades as they can recover exhaust energy from a gas turbine to generate electricity in a Rankine cycle. These systems can increase the efficiency while reducing greenhouse gas emissions. In CCPPs energy is recovered in a heat exchanger called a heat recovery steam generator which works as a connector between the topping cycle (i.e., Bryton cycle) and the bottoming cycle (i.e. Rankine cycle). Therefore, the better design of an HRSG will result in a better performance of the power plant [1,2]. Besides the thermodynamic optimization of the HRSG, the fluid flow in HRSG is another important factor affecting the system performance. The inlet side of HRSG is connected to the exit side of gas turbine by an interconnecting duct. There are many aspects to design HRSG channel, in which the flow correction at inlet duct to provide the uniform distribution of velocity is an important one. The non-uniformity in the gas flow pattern may cause a male heat transfer, which leads to heterogeneous thermal absorption in different positions of tubes. This is hence the flow quality is an effective factor on the overall amount of heat transfer. Velocity profile and flow pattern while passing the inlet duct determine the quality of the flow field in the HRSG casing. In the ideal state, flow should enter the HRSG casing uniformly but in real cases, velocity and temperature profiles vary in the casing entrance section and are not equal in all points. Due to the difference in the amount of receiving gas, upper and lower half of the tube bundles have different temperatures. Therefore, while the lower half temperature rises, upper half will have less temperature. These
temperature and velocity non-uniformities lead to some problems. To assure a desirable velocity distribution an experimental test on the model HRSG duct should be conducted. Therefore, flow patterns have to be analyzed and transition zone should be carefully designed for better uniformity.

The optimum design of HRSG has a particular interest to enhance the performance of heat recovery to raise the effectiveness of the combined cycle. There are several studies on thermodynamic optimization of HRSGs [3–5].

Mohagheghi and Shayegan [6] introduced a new method for modeling a steam cycle in advanced combined cycle by organizing non-linear equations and their simultaneous solutions by use of the hybrid Newton methods. Hajabdollahi et al. [7] modeled an HRSG with a typical geometry and a number of pressure levels used at combined cycle power plants. They also applied a fast and elitist non-dominated sorting genetic algorithm with continuous and discrete variables to obtain maximum exergy efficiency with minimum total annual cost per produced steam exergy as a two objective functions. Ameri et al. [8] conducted the exergy analysis of a combined cycle power plant equipped with supplementary firing (SF) in the HRSG inlet. The results showed that using SF results in an increase in output power while reducing the exergy efficiency of the cycle. In another study Ameri and Ahmadi [9] studied the effect of ambient temperature on exergy losses of a heat recovery steam generator and concluded that ambient temperature will affect the HRSG performance. Ghazi et al. [10] performed a comprehensive thermodynamic modeling of a dual pressure combined cycle power plant and carried out an optimization study to find the best design parameters. Sindareh et al. [11] studied the thermodynamic modeling based optimization for thermal systems in heat recovery steam generator during cold start-up operation. Duran et al. [12] performed a methodology for the geometric design of heat recovery steam generators applying genetic algorithms.

Several researches have also been conducted on the flow pattern inside the HRSG. Yoo et al. [13] considered the change of inlet duct shape for the improved uniform flow and showed its possibility. Ameri and Jazini [14] considered the CFD modeling of heat recovery steam generator inlet duct. In their study the abilities of computational fluid dynamics have been assessed to obtain the crucial profiles without the experimental difficulties. Regarding the special characteristics of flow and geometry, numerical solution may not be performed without taking some techniques into the CFD modeling. Hegde et al. [15] made a modification in the internal configuration of the HRSG. They investigated the influence of a flow correction device (FCD) on the profile of the gas flow entering a high pressure superheater. They modeled the FCD by a perforated plate of a given open area. They also attained a further

### Nomenclature

- $A_1$: first area [m$^2$]
- $A_2$: transition zone [m$^2$]
- $F_g$: body force
- $N$: number of data
- $Q$: flow rate [m$^3$/min]
- $Re$: Reynolds number
- $u$: local normal velocity [m/s]
- $u_{rms}$: RMS velocity [m/s]
- $U$: mean velocity [m/s]
- $V$: velocity [m/s]
- $\theta$: angle [°]
- $\rho$: density [kg/m$^3$]

### Table 1

<table>
<thead>
<tr>
<th>Tests numbers</th>
<th>$Q$ [m$^3$/min]</th>
<th>$A_1$</th>
<th>$A_2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Test 1</td>
<td>2.2</td>
<td>3.86</td>
<td>0.75</td>
</tr>
<tr>
<td>Test 2</td>
<td>3.5</td>
<td>6.14</td>
<td>1.19</td>
</tr>
<tr>
<td>Test 3</td>
<td>5</td>
<td>8.77</td>
<td>1.71</td>
</tr>
<tr>
<td>Test 4</td>
<td>6.5</td>
<td>11.4</td>
<td>2.23</td>
</tr>
</tbody>
</table>

**Fig. 1.** Schematic of horizontal HRSG and fan.
uniform flow distribution through moving the perforated plate to a better location in an optimizing process. Lee et al. [16] investigated the effect of swirl flow of the gas turbine exhaust gas in the inlet duct of a horizontal HRSG. They found that the flow correction devices made the flow entering the duct burner more uniform and the flow uniformity was not affected by the intensity of swirl.

Table 2
The range of inlet duct parameters.

<table>
<thead>
<tr>
<th>Range of length l (mm)</th>
<th>Range of angle δ (°)</th>
</tr>
</thead>
<tbody>
<tr>
<td>135</td>
<td>18</td>
</tr>
<tr>
<td>155</td>
<td>20</td>
</tr>
<tr>
<td>180</td>
<td>23</td>
</tr>
<tr>
<td>200</td>
<td>26</td>
</tr>
<tr>
<td>220</td>
<td>30</td>
</tr>
<tr>
<td></td>
<td>41</td>
</tr>
</tbody>
</table>

Table 3
Conditions of fan for pressure test.

<table>
<thead>
<tr>
<th>Pressure test no.</th>
<th>Percent of inlet voltage of fan</th>
<th>Q (m³/min)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5</td>
<td>2.2</td>
</tr>
<tr>
<td>2</td>
<td>15</td>
<td>3.5</td>
</tr>
<tr>
<td>3</td>
<td>30</td>
<td>5</td>
</tr>
<tr>
<td>4</td>
<td>50</td>
<td>6.5</td>
</tr>
</tbody>
</table>

Fig. 2. Schematic of inlet duct mechanism.

Fig. 3. Velocity profiles of each optimum model at different angles on section A2.

Fig. 4. RMS deviation of 23° of inlet duct at different length.

Fig. 5. Evaluation of mean velocity deviation in numerical simulation.
when the flow correction device was in place. Shin et al. \cite{17} investigated the flow pattern in a complex inlet duct of a heat recovery steam generator. Numerical simulation and experimental study were conducted in order to analyze flow patterns and finally to find out appropriate alternatives for uniform flow in the transition zone of the vertical type HRSG for a gas turbine combined cycle. Flow measurement was conducted in a 1/20 scale-down model. Velocities were measured by pitot tubes on selected sections and uniformity of the flow was quantitatively evaluated. The water table test was also carried out to visualize the flow and to compare the flow patterns according to the selection of alternative shape design of the transition zone, which would lead to better uniformity of the flow in the duct. Patil et al. \cite{18} studied gas flow behavior in HRSG inlet duct with CFD tools. The main aim of the work was to get a maximum uniform flow of flue gases through the inlet duct of the HRSG. The model was developed to simulate the flow of flue gases through the duct by using ICEM software and flow analysis was done by using CFD commercial code FLUENT. The distribution plates in the ducts were added in order to optimize pressure drop and to reduce turbulence of gases caused due to the high velocity of the flow. The focus of the study was to improve the flow characteristics and to make the flow uniform. Hanafizadeh et al. \cite{19} studied the inlet duct of HRSG by part elimination and lattice search (PELS) method to improve the geometry. The RMS (Root Mean Square) difference of local velocity from average velocity is used to evaluate the uniformity of the flow. The terms of uniform velocity and temperature distribution along the HRSG is numerically solved and explained.

This paper is detailed design and experimental work of Ref. \cite{19} published by our research team. The objective of this work is to gain optimum distribution of inlet duct of pilot model of horizontal type HRSG for special operating circumstance. To achieve this, both experimental study and comprehensive numerical simulation are conducted in order to investigate the flow pattern in the transition zone.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{fig6}
\caption{Variations of RMS in optimum length of each angle.}
\end{figure}

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{fig7}
\caption{Variations of RMS in $\theta = 23^\circ$ at different length.}
\end{figure}
zone. Several changes for the best uniformity of flow were performed by quantifiable performance parameters with scale-down model test.

2. Experimental analysis

In this research paper, to investigate the optimum model of HRSG from flow uniformity of inlet duct point of view a 1/10 scale model of the 4 MW semi industrial gas turbine is prototyped. Schematic of HRSG model is shown in Fig. 1. As it is illustrated in this figure, the HRSG entrance is coupled to the fan. This fan has been used to evaluate hydrodynamic modeling and can provide the proportional range of flow rates. The geometry of the model consists of (1 m × 0.8 m × 0.125 m) and the length of inlet duct is 0.3 m (the transition zone height is 0.4 m).

The Reynolds number of the model is anticipated in the range of $2 \times 10^4$–$3.6 \times 10^5$ at the entrance and the surface of the tube zone area and the Reynolds number of prototypes is $6 \times 10^5$. The Reynolds number is similar between prototype and model. It is well known that the turbulent flow pattern does not change any more by increasing the Reynolds number larger than $1 \times 10^4$ [20]. In Table 1 the characteristic of the system according to the operation conditions of fan is shown. The model is able to easily change specific parameters influencing the HRSG. Therefore, cold test is performed to evaluate the flow patterns in HRSG and quantities of non-uniformity.

The number of measurement points was decided to be 11 at $A_2$ to investigate the effect of angles by dividing $A_2$ section to four regions consist of some points (upper, middle, lower plate and main stream of flow). Anemometer device was used for the velocity measurement.
3. Numerical simulation

Numerical method is also conducted to act as a supplement of the experimental investigation. To visualize the flow in the inlet duct and hence to compare the simulation of flow pattern with the experimental data, a 2D numerical simulation is conducted in a 1/10 scale down model. In addition, 2-D model tests have the advantage of being simple to build and easy to change the test conditions. The aim of the changes is to distinguish how the form of inlet duct would affect performance of flow after duct changes on transition zone. Therefore, several cases which have diverse angles and lengths, consist of $\theta$ and $L$ were handled, as have been shown in Fig. 2 and have been summarized in Table 2. The computational domain is limited from the end of the gas turbine to the HRSG outlet. The main stack of HRSG is also included in the domain. One of the major turbulence models is $k-\epsilon$ model which has been implemented in most general-purpose codes, therefore the realizabled $k-\epsilon$ model has been chosen for the present numerical simulation. Numbers of cells vary from 100 to 300 thousand in three values as per geometry dimensions and requirement for a grid independent solution. The numerical simulation has been done based on commercial CFD package (Fluent V6) and boundary conditions, grid convergence, etc. are according to Ref. [19].

3.1. Governing equations

The time-averaged equations for continuity, and momentum are given as Eqs. (1) and (2), respectively.

$$\nabla \cdot (\rho \mathbf{V}) = 0$$  \hspace{1cm} (1)

$$\rho(\mathbf{V} \cdot \nabla)\mathbf{V} = -\nabla P + \nabla \cdot \sigma + \mathbf{F}_b$$  \hspace{1cm} (2)

where $F_b$ is body force, $\sigma$ is stress tensor and $P$ is static pressure.

Navier–Stocks equations are exact equations defining turbulent flows. Analytical approach to solve these equations is available. On
the other hand, to obtain details of small turbulences by solving these equations in detailed procedure, a professional hardware and also much time is required.

From the cold flow tests, the velocity distributions of the different points are obtained and the flow uniformity is analyzed by quantifiable performance parameters. Root Mean Square (RMS) deviation is used to evaluate the uniformity of the flow. Therefore, the least RMS deviation amount is favorable, so that the coordinates of the point matching the least RMS deviation will make the desired shape of the duct. The equations used for the calculation of RMS are as below.

\[
RMS_{\text{deviation}} = \frac{U_{\text{rms}}}{U} \times 100\% 
\]  \hspace{1cm} (3)

\[
U_{\text{rms}} = \sqrt{\frac{\sum (u - U)^2}{N}} 
\]  \hspace{1cm} (4)

In order to measure the flow velocity in the experimental tests, an anemometer device is used. The anemometer is located in the center of the transition zone area of the duct. The test procedure is consisted of six evaluating angles of the inlet duct. At each angle there are five changing lengths that modify the angle of the second plate of inlet duct matching Fig. 2. After preparation of each at specified length, four tests were achieved in the system by various mass flow rates of inlet air to evaluate performance of models at different loads. After performing these steps and evaluate the data, the best length at each angle was selected and was compared based on Root Mean Square deviation of velocity.

In the next step, the optimum models in each angle are investigated according to pressure loss in the transition zone. Low pressure loss means the better shape of inlet duct from flow point of view. In Table 3 input values of fan’s flow rate to express the pressure analysis of optimum model in each angle were summarized.
4. Results and discussion

4.1. Numerical results

The numerical analysis is performed and the amount of the RMS deviation for all angles is studied and reported. The optimal length was selected by comparing the results. In Fig. 3 velocity profiles of the optimum models have been shown. Comparing RMS among different models represents that the angle of 23° has a smoother velocity profile and minimum deviation of mean velocity. In fact, creating an optimal curve of duct changes makes flow possible to adapt to the sudden expansion and make minimum backflow in the higher altitude of the transition zone. Also in Fig. 4 the results of mean velocity deviation have been compared in the angle of 23° in various lengths. It can be seen that by applying the length of 200 mm, length of flow adhesion increases and in fact the flow separation has been postpones to downstream of the inlet duct. Actually, this length is the optimum length and the results show that increase in length may cause an increase again in amount of RMS.

In addition, in Fig. 5 evaluations of mean velocity deviation in numerical simulation are compared together in the range of angles 16–41° for every one degree changes of angle along their optimal lengths and the effect of 23° is quite evident in improving the uniformity of the velocity profile. Briefly, at an angle of 18° the minimum deviation has been allocated to length 200 mm. Also in 20, 23, 26 and 30° minimum value of deviation has been appropriated to a length of 180, 200, 180 and 180 respectively.

Of course, to obtain the exact value of the optimum angle by comparing to various models consist of 22 and 24°, the range of appropriate angle of inlet duct is obtained with a focus on 23°. And also has been detected proper length at an angle of 23 in the range of 200–205 mm.
4.2. Experimental results

In the experimental analysis in this paper, the different models are conducted by four mass flow rate tests. These four tests are performed to investigate various load conditions to simulate the partial load behavior of the system. By changing the input voltage of the fan, the value of mass flow rates is different (as shown in Table 1). To calculate the velocity at different data point and the optimum model from the empirical aspect concludes by comparing of velocity profile from a uniform distribution and the backflow area point of view. The experimental analysis also has been carried out based on the range of numerical angle and lengths. The results show again the angle of $23^\circ$ has the minimum value of the mean velocity deviation and maximum uniformity. Fig. 6 shows the variations of RMS in optimum length of each angle at various mass flow rates. Also Fig. 7 shows the parameter of optimum length in $23^\circ$. In Fig. 8 velocity profiles has been shown in experimental method at $\theta = 23^\circ$. As it can be seen, the amount of velocity declines by the increasing mass flow rate in two regions (positive and negative velocity).

4.2.1. Repeatability

To ensure the independency of obtained results in the same conditions and methods of testing, each measurement has been repeated at least three times. Finally the average of obtained results from different tests was considered as the amount of velocity at the desired point for post processing and computations. Fig. 9 represents the repeatability of test no. 4 in three measurement procedures in the same conditions.
4.3. Pressure test

The target of pressure test is the evaluation of the effect of changing the inlet duct geometry on pressure drop of flow in an HRSG. The lower the pressure drop, the better optimal values for angle and length are desired. Figs. 10–13 show the pressure loss at different models for three different points on the transition zone (i.e., $\gamma = 0.175, 0.525, 0.85$).

It is understandable in all figures that an increase in flow uniformity by applying the optimum model results in lower pressure difference.

4.4. Comparison of experimental and numerical results

In Fig. 14 velocity profiles at optimum angle is compared between experimental and numerical results. As it can be seen here, the velocity profiles have a reasonable match with each other. The location of change in velocity direction in numerical method occurs at 0.21 meter and this height is 0.017 m above for experimental analysis. These numbers are almost close together. The maximum velocity difference between two methods has been occurred near points that velocity direction changes.

4.5. Smoke test

After analyzing the velocity, pressure and determining the optimal geometry, comparison of this model can be shown the diversity between optimum model and initial inlet duct model (without fracture angle) from the hydrodynamics point of view in smoke test. In Fig. 15 smoke test section is indicated. Figs. 16 and 17 show the smoke flow changes inside the body in the optimized and primary structure. Fig. 18 shows when flow arrives due to sudden expansion of duct, the flow separates very quickly from the wall and direct flow has been formed in the lower level of height. Also vortex formation region will be larger. But in the optimum model due to the gradual expansion is associated with duct and the separation time from wall increases and flow be driven to the higher regions. So the vortex area occurs smaller than primary model that its effect is creating more uniformity in the transition zone.

5. Discussion

Case studies by experimental investigation and numerical simulation are conducted to show the effect of inlet duct shape on the flow uniformity and pressure loss. Shape and geometry of the inlet duct will certainly affect the flow behavior. It can produce vortices and secondary flow, which results in non-uniform flow. Even if the change in channel cross section is not gradual the flow separation occurs as a result, it can be seen the optimal model is independent of flow rate and is constant. The upper wall was divided into two parts and the initial angle divided in the five different models, each model includes five lengths in experimental test and wide range angles in numerical methods to get the appropriate size of duct. Also, it has other problems such as the installation difficulty and the possible occurrence of breaking off from inlet duct during operation. Non-uniform flow arising in the transition zone became clear through the investigation by experimental and numerical studies. It can influence the performance and structural damages. Thus, inlet duct should be designed with the consideration of the flow characteristics can also be used as one option for the problem.

6. Conclusions

In this research paper both experimental and numerical studies were conducted to understand the flow patterns in the horizontal type of HRSG’s model. In the cold air test, flow velocity was quantitatively measured with an anemometer to consider the flow uniformity. The alternative design of the inlet duct shape on the flow behavior was also investigated through flow visualization. The recirculation flow zone is noticeable near the sharply turning corners, and this non-uniformity can be eliminated by modifying the geometry of the flow passage. Numerical simulation results were compared to that of the model tests. Experimental analysis is conducted based on uniformity and minimum deviation of the velocity profile by velocity meter device. Also, to investigate the effect of inlet duct geometry, pressure analysis is done between the outlet of the fan and transition zones in different mass flow rates according to various angles and lengths. From this comparison, computational results showed good agreement with the experimental data, and which was considered to be one example of validation.
Optimum design of inlet duct shape is evaluated to reduce the non-uniformity of the flow in the HRSG. Quantitative comparison based on the RMS deviation of the normalized velocity was presented. Among the ideas, 23° of inlet duct plate and 200 mm length showed the best performance for the uniformity improvement. Experimental measurements on the scale-down model were presented as one way of back up to the numerical simulation of the optimum design hydrodynamically. Moreover, the pressure analysis was done between the outlet of the fan and transition zone, which results showed that the optimum design model has the minimum pressure loss.

Acknowledgements

The authors would like to express their thanks to the Center of Excellence in Design and Optimization of Energy Systems (CEDOES), School of Mechanical Engineering, College of Engineering, University of Tehran, for its financial support through the research implementation.

Reference