Optimization of Diverter Box Configuration in a V94.2 Gas Turbine Exhaust System using Numerical Simulation

A. Mohajer, A. Noroozi, and S. Norouzi

Abstract**—**The bypass exhaust system of a 160 MW combined cycle has been modeled and analyzed using numerical simulation in 2D prospective. Analysis was carried out using the commercial numerical simulation software, FLUENT 6.2. All inputs were based on the technical data gathered from working conditions of a Siemens V94.2 gas turbine, installed in the Yazd power plant. This paper deals with reduction of pressure drop in bypass exhaust system using turning vanes mounted in diverter box in order to alleviate turbulent energy dissipation rate above diverter box. The geometry of such turning vanes has been optimized based on the flow pattern at diverter box inlet. The results show that the use of optimized turning vanes in diverter box can improve the flow pattern and eliminate vortices around sharp edges just before the silencer. Furthermore, this optimization could decrease the pressure drop in bypass exhaust system and leads to higher plant efficiency.

Keywords**—**Numerical simulation; Diverter box; Turning vanes; Exhaust system.

I. Introduction

COMBINED Cycle power plants have become a serious alternative for standard coal- and oil-fired power plants because of their high thermal efficiency, environmentally friendly operation, and short time to construct. The combined cycle plant is an integration of the gas turbine and the steam turbine, combining many of the advantages of the both thermodynamic cycles using a single fuel [1].

In large gas turbine combined cycle power systems that produce more than 200 MW power, the cost of fuel is over 80 percent of their operating expense [2] and any improvement in system efficiency will lead to substantial saving in expenses and pollutant reduction. Because of the high velocity and temperature of the exhaust gases (usually between 400 to 650°C), it is necessary to use an appropriate exhaust system. Among the various aspects that have to be analyzed in a cogeneration and combined cycle plant design, the gas exhaust system design can represent a critical aspect, in particular when a bypass stack, which allows the modulation of heat-to-power generation, is present, since it may influence the entire system working condition [3]. Moreover, an exhaust gas system has to satisfy several requirements and constraints, which include architectural and environmental impact, noise levels, structural resistance and thermodynamic efficiency [3]. For combined cycle power plant, the diverter box is used to direct exhaust gas of the gas turbine either into the waste heat recovery boiler or, when running under open cycle mode, exits directly to the by-pass stack [4].

Numerical simulation of a dual-pressure once-through heat recovery steam generator used in combined-cycle gas and steam turbine power plants was carried out by Engoma et al [5]. Their results showed the influence of the temperature and the mass flow rate of turbine exhaust gas on the steam generator prediction behaviors. Pinelli and Bucci [3] analyzed different stack design configurations in a cogeneration power plant to obtain the most efficient configuration of exhaust system. In the plant considered in their work, the exhaust gas system consisted of a single steel stack through which both the main and the bypassed gas streams were ejected in it. The low Reynolds number k-ε Lam–Bremhorst formulation was used for turbulence modeling [3]. The characteristics of the gas flow in HRSG is far different than in exhaust system, because velocity of gas in much higher in exhaust system rather than HRSG; thus in the present work, we have employed standard k-ε turbulence model to simulate the flow field.

Using turning vanes improves the flow pattern in diverter boxes by tempting the gas stream to flow more aerodynamically and preventing it from creation of large vortices, besides using of such equipment provides a uniform flow over silencer blades. This idea, using turning vanes, has been issued as an US patent in 1993 [6], but there is not any report in technical journals or conferences about the effects of practical utilization of them on the efficiency of power plants. Present research is willing to evaluate the effect of using turning vanes on the performance of bypass exhaust system in a combined cycle or exhaust system in a gas turbine cycle.

II. Problem Description

The model of choice consists of the set of equipment that is located in the bypass exhaust system of a combined cycle plant. This equipment comprises of an inlet cone, diffuser, transition ducts, diverter box, silencers, and bypass stack. The

Ali Mohajer, MSc. in Mechanical Engineering, Tarbioat Modares University, is working in MAPNA, MEEMCo (Private Joint Stock) Iran (phone: +98-261-4602358; fax: +98(261)9208052; e-mail: ali.mhj@gmail.com).

Ali Noroozi, PhD student in Mechanical Engineering, in Tarbiat Modares University, is working as Research and Development Unit Manager in MAPNA-MEEMCo. (e-mail: noroozi@mapnaequipment.com).

Saeid Norouzi, MSc. in Material Science, 2002, The University of Tehran. Bs., 1999, The Ferdowsi University of Mashad. Product Engineering Manager, MAPNA-MEEMCo.(e-mail: s_norouzi@mapnaequipment.com).

This research has been financially supported by MAPNA company.

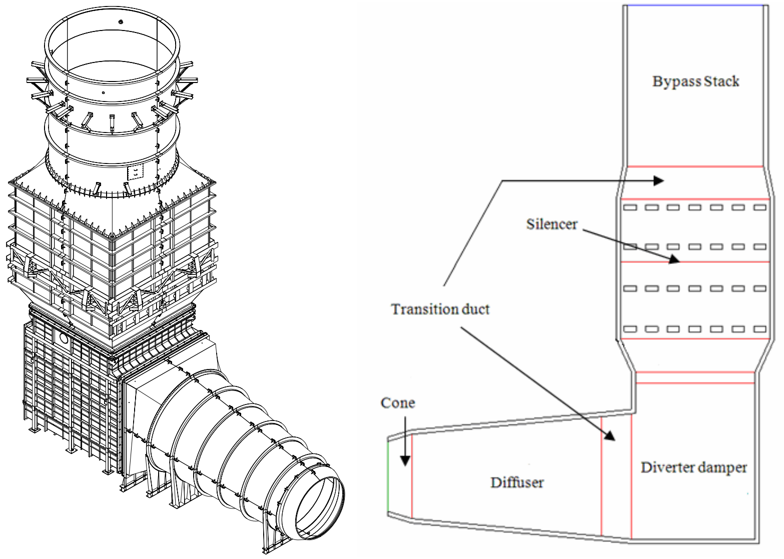

Fig. 1 The 3D drawing (Left) and 2D model (Right) of the exhaust system

geometry and dimensions are based on the Yazd combined cycle power plant units, and the physical properties of exhaust gases and required data related to the flow are in coincidence with the design point of this power plant.

III. Governing Equations and Boundary Conditions

Since gas flow has high temperature, ~841°K, and high velocity, the effect of fluid compressibility must be considered. So, the following preconceptions were considered in the analysis;

- Governing equations consisted of two-dimensional Navier-Stokes equations.
- The flow was considered in steady state condition under the effect of fluid compressibility.
- Because of using isolated walls, all the fluid thermal properties, except density, were considered constant corresponding to working temperature, ~841°K. In addition, the state equation of ideal gas was used to calculate the gas density due to high working temperature.
- Turbulence modeling was carried out via $k-\varepsilon$ standard model.

Navier-Stokes equations in the 2D and steady state condition alongside with energy and continuum equations used in this research are as follows:

$$\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}=0 \tag{1}$$

Momentum equation in the tensor format corresponding to a compressible fluid:

$$\nabla.(\rho.\vec{v}\vec{v})=-\nabla P+\nabla.(\bar{\tau})+\rho\vec{g} \tag{2}$$

Where g is the gravity acceleration, and $\bar{\tau}$ is stress tensor with following equation:

$$\bar{\tau}=\mu\left[(\nabla\vec{v}+\nabla\vec{v}^{T})-\frac{2}{3}\nabla.\vec{v}I\right] \tag{3}$$

Where μ and I are viscosity and unit tensor, respectively.

Primarily simulation revealed that the velocity in some regions of the physical domain increased to some excessive amounts, and the Mach number exceeded 0.3; therefore, it was necessary to add compressibility stress terms to the governing equations. It is especially clear when fluid is squeezing through silencer splitters, which is a kind of geometrical obstacles in front of the flow, causes sudden change in the cross section area. The sound velocity c has been calculated by "(4),":

$$c=\sqrt{\gamma RT} \tag{4}$$

And the density in every element can be calculated from ideal gas relation:

$$\rho=\frac{P_{op}+P}{\frac{R}{MW}T} \tag{5}$$

And the energy equation is as follows:

$$\nabla.(\vec{v}(\rho E+P))=\nabla.(k_{eff}\nabla T) \tag{6}$$

Where k_{eff} is the overall thermal conductivity which is a combination of the turbulent effects and thermal conductivity of the fluid. And the energy term, E, is defined by "(7),":

$$E=h-\frac{P}{\rho}-\frac{v^2}{2} \tag{7}$$

IV. Turbulence Modeling

The $k-\varepsilon$ standard model has been utilized in order to model the turbulence of the flow. k refers to turbulent kinetic energy and the term ε refers to turbulent energy dissipation rate. Among all two-equation turbulence models that have been presented so far, the model proposed by Launder and Spalding is the most well-known, k-ε standard model. In this model, the turbulent viscosity is calculated as following:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{8}$$

Where k and ε were calculated from the following partial differential equations:

$$\rho U_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right) + P_k - \rho\varepsilon + g_i \beta \frac{\mu}{\sigma_T}\frac{\partial T}{\partial x_i} \tag{9}$$

$$\rho U_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}\left(P_k + C_{3\varepsilon} g_i \beta \frac{\mu}{\sigma_T}\frac{\partial T}{\partial x_i}\right) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{10}$$

V. Numerical Approach

In order to numerically simulate the flow field within the domain, the commercial FLUENT 6.2 software was used based on the control-volume approach. The equations of scalar variables were discretized by QUICK scheme. For handling the pressure–velocity coupling, the SIMPLE algorithm was implemented.

It should be noted that there were some serious convergence problems. In order to get to reliable results, we started from lower inlet mass flow rates and then increased it step by step. Every time the convergence was obtained in a specified mass flow rate, before increasing the mass flow rate to a higher level we switched the equations to time dependent conditions and after getting to semi-steady state conditions, we again switched to steady state equations.

VI. Grid Independency

In order to achieve a suitable grid with proper concentration, the selected geometry was meshed using four different sets of grid with different concentration. The grid independency was checked out using the bypass exhaust system model without inserting turning vanes.

According to Fig. 2, which represents the angle of velocity vectors at the inlet cross section of diverter box, the grid with 422691 elements provided the results independent from grid concentration, in the other words, finer mesh did not have any effect on accuracy of the results.

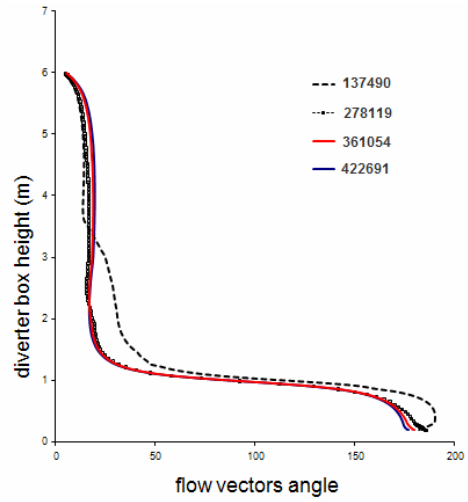

Fig. 2 Grid independency checkout

VII. Results and Discussion

The purpose of this research is investigation of the effect of diverter box turning vanes on the energy loss and pressure reduction in the exhaust system of a combined cycle. In this research, first the current diverter box design which is installed in Yazd power plant was studied, and the flow of exhaust gases through this system was simulated.

The profile of velocity vectors in Fig. 2 represents that the attacking angle of the turning vanes in the inlet section of diverter box should be set at 13 degrees by average so that to provide the vanes with most streamwise conditions. Then simulation of flow field performed in several cases so that the other geometrical factors of the turning vanes were designed based on trial and error approach. The aim of these simulations was to reach to a design that most appropriately fit the idea of eliminating vortices and reducing the intensity of turbulence in the flow field. The results of these simulations showed that the end angle of the first turning vane should be 90 degrees in order to reduce the large vortex hat has been produces at the top sharp edge of diverter box and split it to two smaller vortices.

Also this may have some consequences such as unsuitable flow separation at the back of the vane. This can be seen in Fig. 4. In this figure, it is evident that the especial design of the first vane has reduced the dimension of the vortex just above diverter box at the expense of creating inappropriate exit conditions of the flow. The other conclusion from these simulations is that the exit angle of the other vanes should be a bit less than 90 degrees. This would result in more appropriate distribution of flow through silencer.

Fig. 4 shows that using turning vanes increases the efficiency in terms of increasing engaged cross section area and eliminating large vortices which often form because of sharp geometries. Having better distribution of fluid just before the silencer is another advantage of applying such modifications.

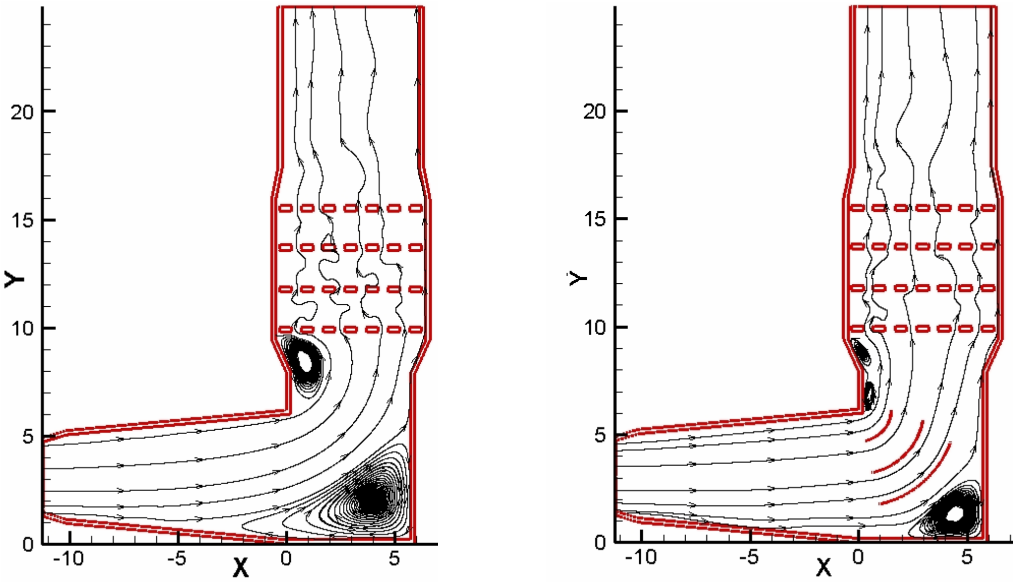

Fig. 3 Flow pattern in exhaust system in two models (dimensions in meters)

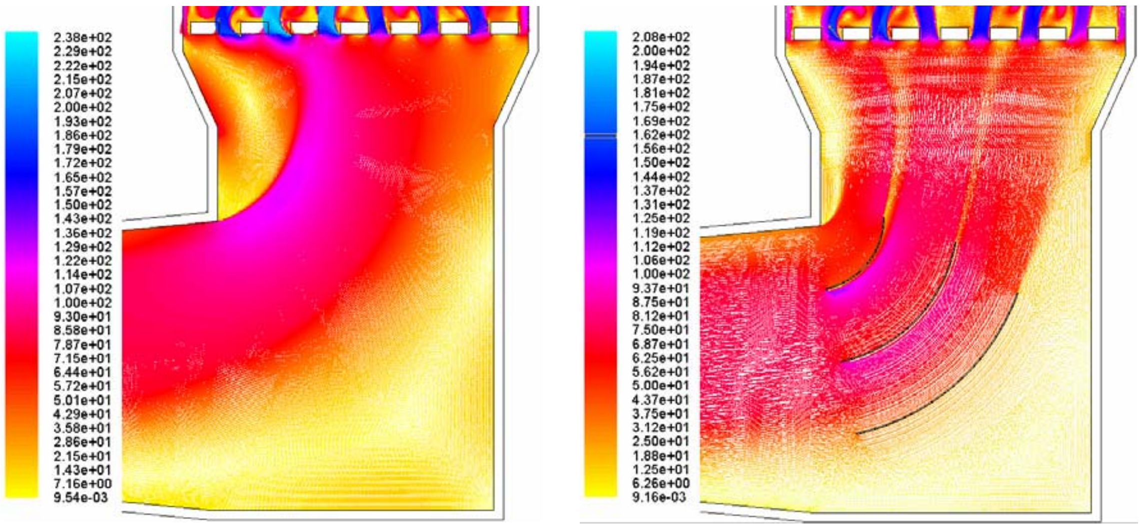

Fig. 4 Comparison of flow pattern in diverter box (velocity vectors m/s)

According to Fig. 5, in the improper design the fluid separation and vortex formation at the outlet section imposes an extra pressure drop.

To evaluate the performance of an exhaust system in a combined cycle system two main criteria, the efficiency of system in terms of pressure drop and noise reduction, are being considered. However, since this system is not used while ordinary service of combined cycle power plants, the environmental issues and reduction of pollutants is not the main concern of designers.

Installation of turning vanes affects the velocity vectors angle profile at the inlet of diverter box, as it reduces the dimension of vortices at the bottom of diverter box. This is shown in Fig. 7 where the height in which the velocity vectors angle is about 180 degrees, in the optimized model is less than that of the present model.

The most important criteria in the evaluation of exhaust system performance in a combined cycle power plant are the efficiency of system in terms of pressure drop and reduction of noise created due to gas flow conditions. Calculations have been done in three different mass flow rates and the results are shown in Fig. 8. According to the graph shown in this figure, using turning vanes causes decreasing in pressure drop; so, gas turbine will work in higher pressure ratio thanks to have the diverter box equipped by turning vanes, which increases gas turbine efficiency.

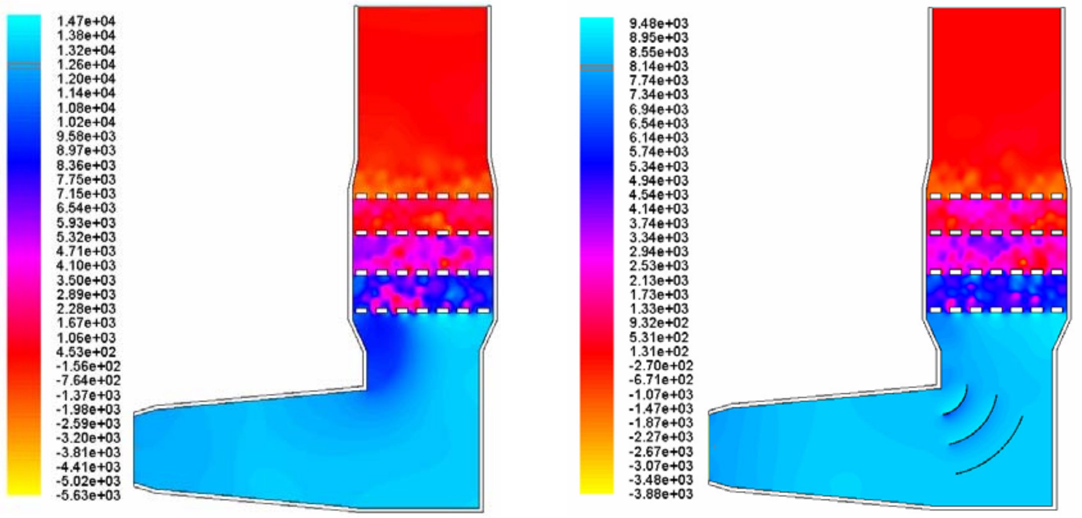

Fig. 5 Comparison of pressure contours in present and optimized model (pa)

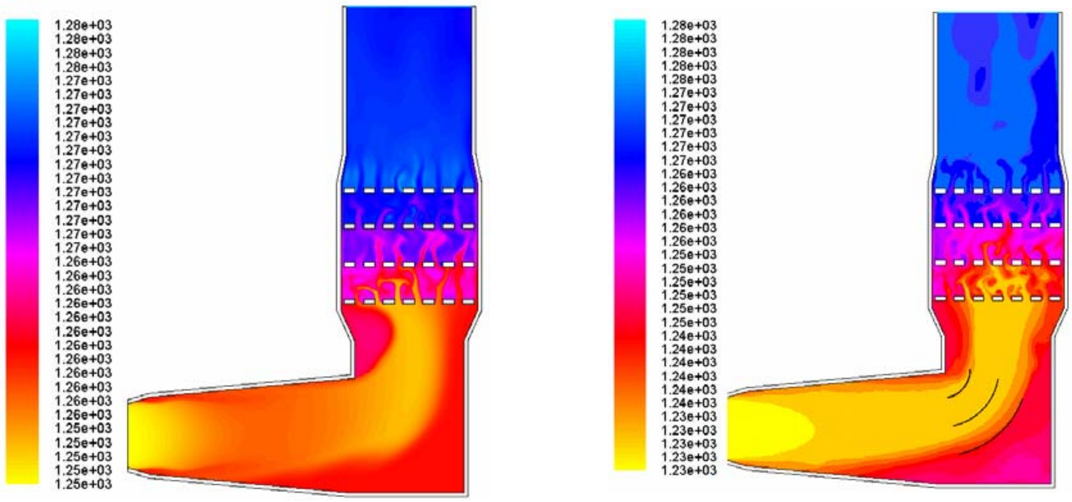

Fig. 6 Comparison of entropy contours in present and optimized model (J/kgK)

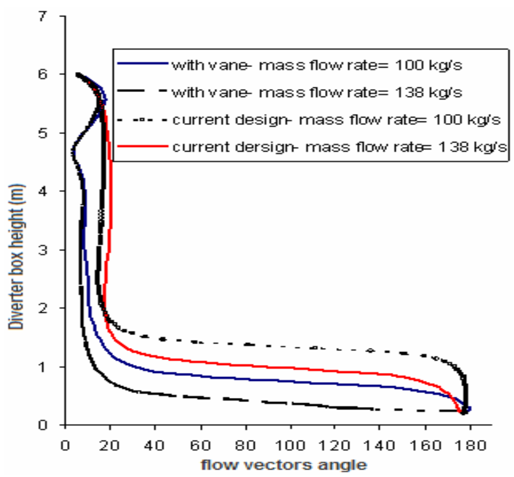

Fig. 7 The velocity angle at the inlet of diverter box

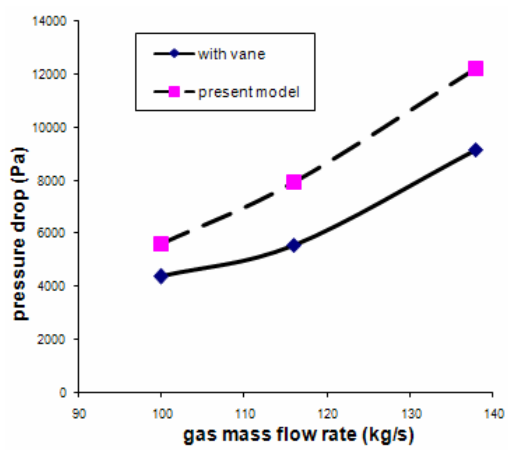

Fig. 8 The influence of using turning vanes on exhaust system pressure drop (pa)

VIII. Conclusion

2D simulation of exhaust gas flow using FLUENT 6.2 software to investigate the effect of implementation of turning vanes in order to modify the flow pattern in diverter box showed that the angle of turning vanes equal to 13 degrees provides the best efficiency regarding pressure drop and energy loss.

Turning vanes increase efficiency due to eliminating of big vortices created in sharp corners, which it has a prominent effect on the modification of flow pattern and on the decreasing energy and pressure dissipation and this leads to higher plant efficiencies.

It should be noted that two dimensional modeling can not involve all the details of the considered system and it may cause some exaggerations in pressure drop in the system because of neglecting bypass paths for the flow encountering obstacles, particularly in silencer splitters. Also, since there was not any experimental data in fingertips, verification of the results obtained by numerical simulation could not be performed. Therefore, these results could be as an estimation of what occurs in reality.

Acknowledgment

The authors wish to thank the financial support of the MAPNA group and MEEMCo executive management to provide writers with the opportunity to perform the research.

References

[1] Sunil Kumar Vylta VV, Huang GPG., “CFD modeling of heat recovery steam generators using fluent,” *30th annual dayton-cincinnati aerospace science symposium*; 2000.

[2] R. G. Surette, “Feasibility Analysis and Final Report: Methods of Improving Efficiency of Combined Cycle Power Plants,” *California Energy Commision – Energy Innovations Small Grant Program*, May 2005.

[3] M. Pinelli, G. Bucci, “Numerical based design of exhaust gas system in a cogeneration power plant,” *International Journal of Applied Energy* (2008) 1503-1512.

[4] Mohd. Zamri Yusoff, Zainul Asri Mamat, Leong Chee Wai, Ibrahim Hussein, Mohd Hariffin Bosrooh, “Analysis of Flow Induced Vibration in Combined Cycle Diverter Damper,” *Proceedings of the Brunei International Conference on Engineering and Technology (BICET2001)*, Bandar Sri Bagawan, Brunei, October 2001.

[5] G. D. Ngoma, A. Sadiki, R. Wamkeue, “Efficient Approach in Modelling and Simulation of Dual Pressure Once-through Heat Recovery Steam Generator,” *Proceeding (379) Power and Energy Systems* – 2003.

[6] William F. Koch - Wahlco Engineered Products, Inc., Lewiston, Me., “Gas Flow Diverter,” *USpatent* 5299601 (1993).