The CFD Modeling of Heat Recovery Steam Generator Inlet Duct

Mohammad Ameri*, Farnaz Jazini Dorcheh

Combined Heat & Power Specialized Unit (CHP), Mechanical & Energy Engineering Department, Power and Water University of Technology

P.O. Box 16765-1719, Tehran, I.R. Iran

*ameri_m@yahoo.com

Abstract*-The axial velocity and temperature distributions at the outlet section of inlet diffuser of Heat Recovery Steam Generator (HRSG) channel should be uniform as much as possible to avoid overheating of first rows of boiler heat exchangers tubes. Due to flow properties and angle of inlet diffuser, providing a uniform outlet velocity profile is impossible without using a correction device. A proposed design should be checked to satisfy the outlet velocity and temperature requirements. In current 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. The actual HRSG inlet channel incorporates one perforated plate to correct the flow and three burner elements inside its wide-angle diffuser. Investigations have shown that the perforated plate and heat exchanger modules can be modeled by porous jump boundary condition and the burner elements by radiator faces respectively. Realizable k-ε with non-equilibrium wall function seems to be the most optimum turbulence model for solution of the problem.

Keywords- HRSG; Inlet Duct; Flow Correction; Diffuser; CFD

I. INTRODUCTION

The Power generation by application of combined cycle using HRSG is widely and progressively used throughout the world. In addition to power generation, heat recovery boilers may be used to supply process steam for many applications.

There are many aspects to design HRSG channel, among them is the flow correction at inlet duct to provide the uniform distribution of velocity and temperature in sections such as diffuser outlet and burners location. The most common device to expand the jet flow at inlet diffuser on the cross-section is the perforated plate. The resultant flow pattern depends greatly on the geometry and location of perforated plate. To assure a desirable velocity distribution an experimental test on the model HRSG duct should be conducted and final choice of perforated plate will be achieved by a type of trial and error procedure for different perforated plate type and displacement of it either forward or backward in the diffuser section.

Since the numerical modeling has been always more flexible and with less costly than the experimental approach, therefore it will be preferred for flow correction in HRSG if there are some techniques to overcome the difficulties related to CFD modeling of the HRSG Duct. High Reynolds number and swirling flow, huge dimensions, existence of secondary cross flows in rectangular sections, wide-angle pyramidal inlet diffuser, a big outlet stack, many tube rows of heat exchanger, perforated plate with its holes and baffles and burners, can be mentioned as obstacles in CFD modeling of HRSG. The goal of this paper is to give some guidelines and obtain the flow field using CFD in an actual HRSG inlet duct with supplementary firing.

II. GEOMETRIC MODELING

As claimed by Lee et al.[1], the main stack at the end of the channel has almost no influence on the inlet duct flow field of HRSG. It may be due to presence of the heat exchangers which damp the vertical pressure gradient formed by stack. Fortunately the most important problem in HRSG duct meshing is related to last stack because it is not consistent by upstream rectangular cross-section to make a structured grid. Removing the stack reduces the number of cells greatly. In fact, in this paper, removing this 25 m tailpipe with 5.4 m diameter has improved convergence a great deal. Figure 1 shows the real dimensions of HRSG duct configuration which has been studied by CFD.

There is no need to build exactly the circular to rectangular entrance transition duct. The transition length usually is very small compared to other dimensions. Therefore, the whole entrance channel can be modeled by a type of pyramid which has a circular inlet and rectangular outlet; otherwise there will be a divergence of solution.

Every heat exchanger module can be modeled by a porous jump face. Porous jump is built by a sink term which is added to momentum equation of the main flow. In fact, as the Reynolds number is usually of the order of 10^6, this sink term is only made by inertial losses due to the fins. No resistance is considered in other directions. This feature helps to stability and convergence of solution.

DOI: 10.5963/IJEE0303003

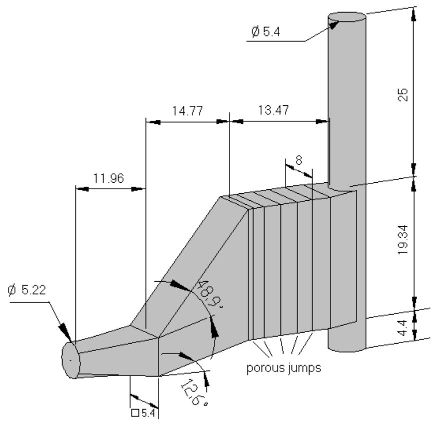


Fig. 1 Geometry of HRSG duct which is studied byCFD (Dimensions in meter)

Lee et al.[1] also used perforated plates instead of heat exchangers in their experimental model. They considered no need for exact modeling of fins. It was due to the fact that the fins' pressure resistance influences the upstream the flow field at the inlet duct. The loss coefficient of every porous jump should be determined by a trial and error method such that it makes the same pressure drop as its related module. The comparison of 3D and 2D models has shown that the trial and error method to find the pressure loss coefficients can be done more rapidly by 2D projected model instead of the actual geometry.

III. PERFORATED PLATE & BURNERS MODELING

The perforated plate was also modeled by porous jump. To verify this simulation, it was decided to answer the question as to what extent the CFD is capable of solving the problem of flow correction in diffuser using perforated plate. It was tried to obtain the experimental results of Ward-Smith et al.[2] who conducted a test on a wide-angle pyramidal diffuser and obtained the properties and locations of two perforated plates which provide a uniform flow in specified downstream section. It was revealed that after the examination of various numerical models that the porous jump model for the perforated plate has the best compatibility and convergence. In fact, the CFD modeling is capable of predicting the downstream velocity distribution if axial jet flow has not been established. After incorporating the perforated plate in diffuser, jet flow will be removed and the numerical results will be reliable.

Fig. 2 shows the front view of burners and baffle solid walls between them. The baffles have been used to concentrate the flow on relatively small burners. Each burner's element mouth is 13.3 cm which is very small relative to section height. Therefore, each burner element can be replaced by a face with radiator condition. The radiator face releases a specified rate of heat which is swept to downstream when the flow passes through it. This model is used when the rate of heat production is known.

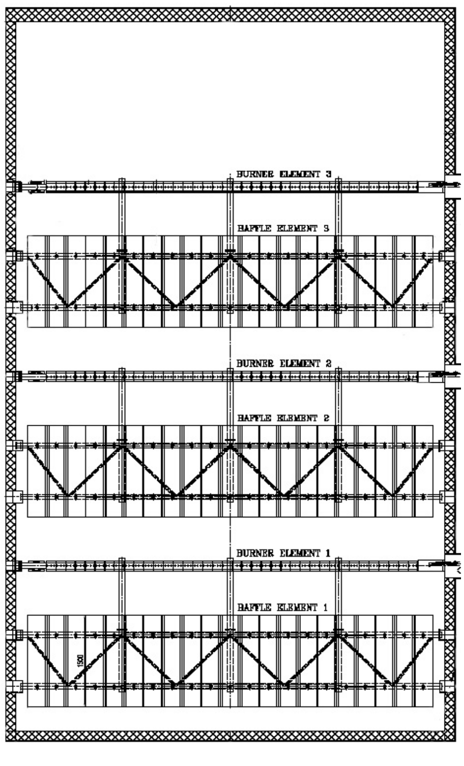


Fig. 2 The front view of burner elements and baffle solid wall

DOI: 10.5963/IJEE0303003

IV. TURBULENCE MODELING

The duct cross-section is rectangular and the flow has some degree of swirl. Therefore, for capturing the flow in separation zone, one of advanced models of turbulence should be used. As there are a huge number of cells, therefore it is better to choose a model with fewer equations. The comparison of the results of Reynolds stress model and Realizable k-ε has shown a negligible difference. Therefore, it seems that Realizable k-ε is the best choice as this model has shown a powerful stability than the others to solve the problem of diffuser-perforated plate combination.

In order to consider the axial flow pressure gradient, the non-equilibrium wall function should be used. The value of dimensionless parameter y^+ represents the validity of wall function application. It must be at most between 20 and 500; otherwise the solution is not reliable. It must be noted that the height of first layer of mesh adjacent to the wall and the Reynolds number, determines the value of y^+ and the validity of solution. The larger Reynolds number requires more refined mesh. Since the successive ratio of cells should not exceed 1.05, decreasing the first layer height involves increasing the total number of cells. The maximum amount of cells depends on the available computer resources. Nevertheless, it is a restricted value especially for the current computers.

V. SWIRL MODELING

In the current case, there was not available any detailed data about inlet velocity profile. The problem is solved for the maximum swirl angle equal to 25°, with forced vortex profile for the inlet swirl velocity.

The swirl number is defined as the ratio of angular momentum flux to the axial momentum flux. It is equal to the tangent of swirl angle in the forced vortex and can be assumed as angular velocity in the axial direction.

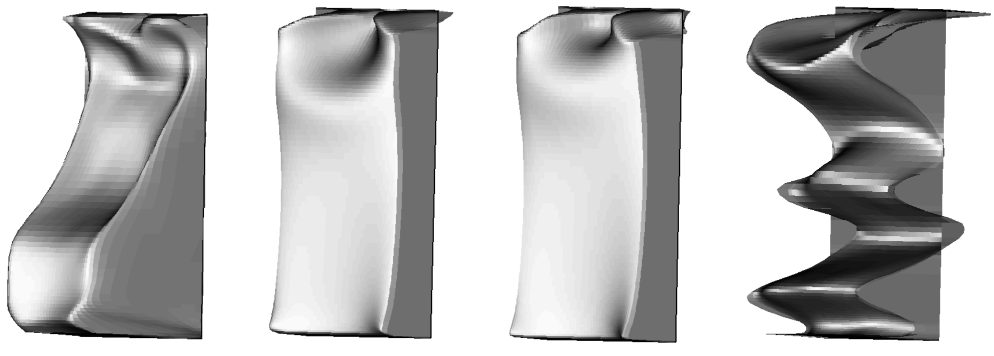
VI. PROMOTING CONVERGENCE

It is very important that one should not begin with empty duct and uniform inlet flow. As a matter of fact, it is not the simplest case and one may never reach the solution for actual Reynolds number. Porous jump faces related to the heat exchangers cause the flow to move upward and prevent the concentration of flow in the bottom of duct. Therefore, they reduce the velocity in cell adjacent the bottom wall and decreases y^+ .

In current study, the presence of swirl and high temperature will tend to improve the convergence of solution. The swirl postpones the separation [3]. Also, an increase in the temperature causes the kinematic viscosity of exhaust gas to increase; which tends to reduce the Reynolds number.

VII. RESULTS AND DISCUSSION

In current study, the values of perforated plate pressure loss coefficient and the porosity are suggested in order to keep its location unchanged with respect to the actual design. Due to the existence of baffles large solid walls, the generation of wake behind them is unavoidable. Therefore, the exit velocity profile will not be influenced. The axial velocity profile in the outlet section of the diffuser can be improved by increasing the pressure loss coefficient of perforated plate if there are not any burners and baffles. There is a maximum pressure loss coefficient per thickness equal to 200 which exist for the velocity profile with the most uniformity. By fixing perforated plate coefficient, the baffles can be brought into the model. Therefore, as it will be expected, their wake will destroy the uniformity of the flow. Figure 3 shows improvement of exit velocity profile by the perforated plates and its destruction after entering the baffles.



(a) (b) (c) (d)

Fig. 3 Outlet velocity profile (a) without perforatedplate, (b) & (c) with perforated plate & losscoefficient per thickness of 170 & 200, (d)velocity uniformity destruction by addingbaffles

Fig. 3 (a) shows the velocity profile without any perforated plate. Figs. 3 (b) and (c) present the uniform velocity profiles by using the perforated plates with the pressure loss coefficient per thickness of 170 and 200 respectively. Fig. 3 (d) shows that

DOI: 10.5963/IJEE0303003

the velocity profile uniformity can be destroyed by using baffles. These results are verified for 8×10^5 and 1.2×10^6 cells.

As mentioned before the uniformity of axial velocity is also important in passes between baffles to attain uniform flame lengths. The established velocity distribution between baffles due to the suggested perforated plate is shown in Fig. 4.

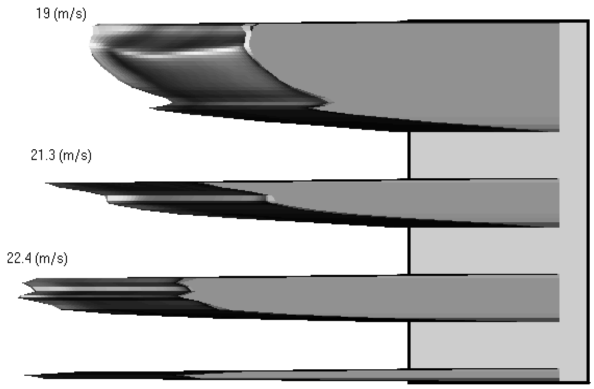


Fig. 2 Velocity profile between the passes of baffles

In fact, this velocity distribution is the most uniform profile which can be obtained by current location of perforated plate and the degree of inlet swirl. The distribution of temperature at the burner section (Fig. 5) shows that the flame lengths seem to be almost uniform.

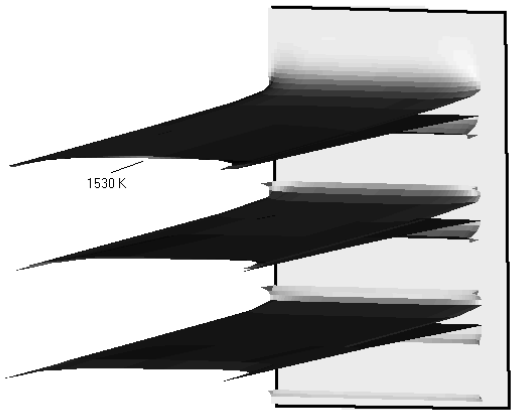


Fig. 3 The temperature profiles at the burner section

As it is indicated in Fig. 6, the temperature profile in the outlet section of diffuser satisfies the standard definition of uniformity i.e. the maximum variation of 20% of the average value in the section for 90% of the area.

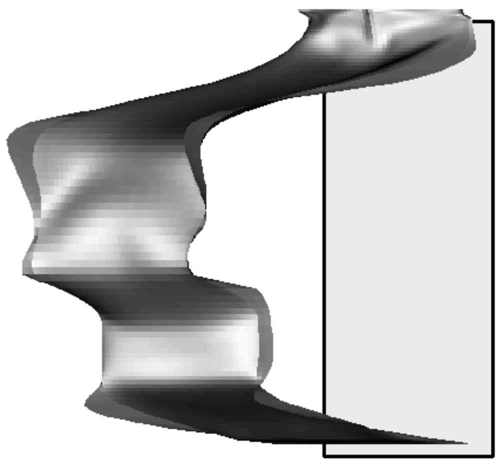


Fig. 4 Temperature profile in exit section of diffuser

For the thickness of perforated plate equal to 15 mm and the value of 1.5 for thickness to the diameter ratio (which is recommended as an optimum value [2]), the suitable porosity and the pressure loss coefficient of the suggested perforated plate may be calculated as it follows.

DOI: 10.5963/IJEE0303003

Pressure loss coefficient per thickness=200

$$K = 200 \times 0.015 = 3$$

If one applies the proposed the empirical relation by Stichlmair et al.[5], one may write:

$$\begin{aligned} K &= K_0 + \phi^2 - 2\phi K_0^{0.5} \qquad \text{for } \frac{t}{d} < 2 \\ K &= K_0 + \phi^2 - 2\phi \qquad \text{for } \frac{t}{d} > 2 \end{aligned} \tag{1}$$

in which $\frac{t}{d}$ is the thickness to the diameter ratio, φ is porosity and $K_0 = 1.55$. Therefore, the corresponding value of φ will be 45%. It should be noted that Equation (1) is based on the velocity through the holes instead of average velocity at the section which is used in the current study. These two may be converted by using φ.

Since dimensions of cross-section are known, having porosity, thickness and $\frac{t}{d}$ of perforated plate, its geometry can be determined as following (Fig. 7):

$$\begin{aligned} d &= \frac{t}{1.5} = 10mm \\ A_d &= \frac{\pi d^2}{4} = 0.0000785m^2 \\ N &= \frac{0.45 * 64.15}{A_d} = 367739 \end{aligned} \tag{2}$$

$$\frac{p}{p'} = \sin 60 \tag{3}$$

$$\begin{aligned} N &= RC + (C-1)(R-1) \\ R * 2p &= 9.82 \\ C * p' &= 6.53 \\ \frac{R}{C} &= \frac{9.82}{2 * 6.53 * \sin 60} = 0.866 \end{aligned} \tag{4}$$

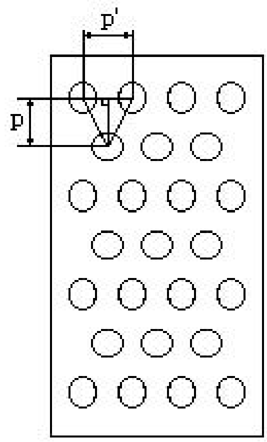


Fig. 5 Triangular arrangement of holes in perforated plate

in which R and C are the number of holes in the first column and first row respectively. One may combine Equations (2), (3) and (4) to get the following results:

DOI: 10.5963/IJEE0303003

$$R = 400$$
$$C = 461$$
$$p' = 14.2mm$$
$$p = 12.3mm$$

VIII. CONCLUSIONS

Regarding to limited time and computer resources, some tricks should be used to solve the problem of inlet duct flow correction in HRSG using computational fluid dynamics.

The inlet duct flow field can be obtained numerically for the actual flow Reynolds number by removing the main stack, incorporating heat exchangers, swirl velocity and high inlet actual temperature. Heat exchanger modules and perforated plates may be modeled by porous jump. Also, each burner element can be modeled by radiator face.

Realizable k-ε with non-equilibrium wall function can bean suggested as optimum turbulence model to estimate the HRSG inlet duct flow field.

Results of numerical solution of HRSG inlet duct have shown that three large baffles destroy the exit velocity uniformity which was produced by suitable perforated plate. However, the flame lengths and temperature distributions have been favorable.

The geometry and properties of the optimum perforated plate to correct the inlet duct flow have been obtained through CFD approach.

The result of this study can be used for HRSG inlet duct design and its optimization.

REFERENCES

[1] B. E. Lee, S. B. Kwon, C. S. Lee, 2002. On the effect of swirl flow of gas turbine exhaust gas in an inlet duct of heat recovery steam generator, ASME Journal of Engineering for Gas Turbines and Power, 124: 496-502.

[2] B. Sahin, A. J. Ward-Smith, 1986. The use of perforated pates to control the flow emerging from a wide-angle diffuser with application to electrostatic precipitator design, International Journal of Heat and Fluid Flow 8 (2): 124-131.

[3] H. Lefebvre, 1983. Gas turbine combustion, Press Roman by Hemisphere publishing.

[4] G. Gan, S. B. Riffat, 1997. Pressure loss characteristics of orifice and perforated plates, Experimental Thermal and Fluid Science, 14: 160-165.

[5] M. J. Lockett, 1986. Distillation tray fundamentals, Cambridge University Press, Cambridge, UK.

Mohammad Ameri was born in Abadan, in southern part of Iran in 1956. He got his B.Sc. in mechanical eng. from Sharif University of Technology, Tehran, Iran in 1980, M.Sc. in thermal & fluid eng. from Amirkabir University of Technology, Tehran, Iran in 1986 and Ph.D. in thermal & fluid eng. of mechanical eng. from Case Western Reserve University, Ohio, USA, 1993.

He has been the associate professor in the Mechanical & Energy Eng. Department at the Power & Water University of Technology, Tehran, Iran, since 1980. He was the UNDP & GEF National Technical Advisor for the Iran Industries Energy Efficiency & Conservation Project. He was the advisor & Project Manager of the "Preparation of Power Enhancement Test Code & Supervision of the Test for the Power Enhancement Projects" for TAVANIR (Iran Power, Transmission & Distribution Co.). He has been recently the National Advisor for Development Study for Institutional Capacity Development on Energy Management in the Building Sector in The Islamic Republic of Iran for Mitsubishi Research Institute (MRI), Japan.

Prof. Ameri has been selected as the distinguished researcher of the country and Tehran Province. He has presented & published more than 120 papers in the international journals and conferences.

Farnaz Jazini Dorcheh was born in Tehran, Iran in 1988. She got her B.Sc. in mechanical eng. from Ferdowsi University, Mashad, Iran in 2004 and her M.Sc. in thermal & fluid eng. from Power & Water University of Technology, Tehran, Iran in 2006. She has worked in Azarab Heavy Industries in Boiler Design Department for several years.

DOI: 10.5963/IJEE0303003