Numerical Investigation of Heat and Fluid Flow Characteristics through Steam Coil Air Preheater Installed in 100 MWe Utility Boiler

E Ariyanto\textsuperscript{1,a}, W A Widodo\textsuperscript{2,b}

\textsuperscript{1}PT Pembangkitan Jawa Bali/UBANG, Jakarta, Indonesia
\textsuperscript{2}Department of Mechanical Engineering, Institut Teknologi Sepuluh Nopember, Surabaya, Indonesia
\textsuperscript{a}eko.ariyanto@ptpjb.com, \textsuperscript{b}wawanaries@me.its.ac.id

Abstract. This paper study the flow characteristic through a set of tube bank used as air preheater in a 100 MWe utility boiler. The study using numerical simulation commercial Computational Fluid Dynamic (CFD) software. A 3D model uses to analyze the flow characteristics with 3 load variations. The model using a steady-state flow assumption. The Steam Coil Air Preheater modelled as a porous medium (PM) to represent its effect on the airflow through it. Porous Medium model can avoid a time consuming mess generation and simulation. This paper using an empirical equation to define inertial resistance coefficient (C2). The model using Heat Exchanger Model, Energy Model, and Realizable k -Epsilon for turbulence viscous model. 100\%, 75\%, and 50\% load variation simulated with the model. The simulation result in both quantitative and qualitative data. The velocity and pressure distribution affected by the load variation, geometrical shape change, and tube bank set. Some secondary flow observed due to geometrical change along with the flow. The heat transfer characteristics observed with temperature distribution show that it affected by the flow characteristic.

1. Introduction
Nowadays, electricity becomes a primary need for human beings, almost all of modern human daily activity using electricity. To produce electricity a Power plant converts prime energy from fuel to electricity. One kind of powerplant is a Steam Turbine Powerplant. A steam turbine powerplant convert fuel energy into steam to rotate turbine and generator to produce electricity.

A 100 MW e Power plant main equipment are Boiler, Steam Turbine, and Generator. Generator converts mechanical rotating energy into electricity, Steam Turbine convert heat and potential energy in the steam to a mechanical energy and Boiler convert Fuel into heat energy in the steam. A Boiler air system provides adequate air to the boiler to burn fuel then heat the water/steam to a certain condition. The air supplied by 2 Forced Draft Fans through ducts to Air Heater. In the Air heater, the air heated to a certain temperature. Then the air goes to the wind box and distributed to each burner. The air heater consists of the main Air Heater and Steam Coil Air Preheater. Main air heater use boiler flue gas to heat the air. Steam Coil Air Preheater heat the ambient air before entering the Main Air Heater.
Heating air before enter the main air heater avoids sulphur condensation in the gas side main air heater.

Steam Coil Air Preheater use steam supplied from auxiliary steam header to heat the air. It’s a cross-flow heat exchanger with steam inside the tube and air outside. It’s consists of 4 row/module of the tube bank across the airflow in the boiler air system. Steam Coil Air Preheater tube bank will increase flow resistance and power for the Forced Draft fan (FDF) to supply air to the boiler.

There are some previous studies that deals with Heat Exchanger modelling using Computational Fluid Dynamic Software. A CFD Modelling can be promising for the design and optimization of heat exchanger it allows testing of numerous design options without fabricating Physical prototype [1]. Many types of heat exchangers and their related issues such as flow maldistribution, fouling, pressure drop, and thermal Analysis can be studied using CFD modelling [2]. Porous Medium modelling (PM) can simplify the modelling process using CFD; it avoided the need to model a real complex geometry of heat exchanger. If some heat exchanger performance data such as pressure drop, heat rejection, and mass flow available; CFD software can model the Heat Exchanger as a Porous Medium (PM) [3]. Wang et al. [4] study show a good result using PM modelling in CFD Software. The model shows a Hydrodynamic characteristic result closed to the experimental one.

Figure 1 shows the steam coil air preheater and its schematic drawing, this study simulates the heat and fluid flow characteristic in the steam coil air preheater with load variation. 3 load variation simulated in this study, the area 100%, 75%, and 50% respectively. The simulation results show heat and fluid flow characteristic such as Temperature, Velocity, and pressure distribution along the Steam Coil Air Preheater. CFD simulation using PM to model the Heat Exchanger using an empirical equation to define inertial resistance coefficient (C2) result data closed to the commissioning data.

![Figure 1. Steam Coil Air Preheater and its schematic drawing](image)

2. Modelling and Simulation

To analyze the heat and fluid flow characteristics, this study using commercial CFD software to analyze heat and fluid flow characteristic inside SCAH with various load condition. The SCAH as heat exchanger modelled a porous medium to avoid real complex geometry modelling, time-consuming and high CPU usage simulation. The inertial resistance coefficient for the porous medium defines using an empirical equation. The steps for Modeling and Simulation are Pre-processing, Processing, Validation and Post Processing.

2.1 Pre-processing

The first work for the simulation is establishing basic geometry of SCAH and Meshing. The tube bank only described as some boxes that represent the tubes bundle. The SCAH consist of 4 modules/tube bank, the 1st and 2nd module each have 80 circular finned tubes in one row and 3rd and 4th module each have 160 circular finned tube in 2 rows in a staggered configuration. The meshing process
divides the flow passage into a small part in the CFD Software. This study using hexahedral map meshing and is shown in figure 2.

![Hexahedral Map Meshing for basic geometry of SCAH](image)

**Figure 2. Hexahedral Map Meshing for basic geometry of SCAH**

### 2.2 Processing

The simulation using commercial CFD software with energy equation and heat exchanger macro. Realizable k-ε with enhanced wall used for viscous parameter. The heat exchanger macro parameters are: fixed heat rejection and simple effectiveness method for heat transfer. Cell Zone Condition heat exchanger modeled as porous medium. Inertial resistance and porosity defined in the porous medium. Inertial resistance defined using empirical equation for inline and staggered tube arrangement. Plant first commissioning provide data to calculate total heat rejection in the SCAH. The total heat rejection divided proportionally with heat transfer area for each module/tube bank. Heat exchange effectiveness needed to simulate using Simple effectiveness heat transfer model. The heat exchanger effectiveness calculated with divided actual heat rejection with maximum possible heat transfer and result 0.224, 0.176, 0.177 for 100%, 75% and 50% load respectively.

Viscous and inertial resistance contributes for pressure drop along the flow. To reflect these factors on PM model, the momentum equation added with a source term \( S_i \).

\[
S_i = -\left( \frac{\mu}{\alpha} \frac{\partial u_i}{\partial x} + C_2 \right) \rho \frac{1}{2} \left| v \right| v_i
\]

When fluid flow is low, Viscous resistance contribute dominantly for pressure loss. For high velocity like the flow in boiler duct, the inertial resistance become dominant and Equation 1 can be rewritten on one direction:

\[
\Delta p = \frac{\rho v^2}{2} x C_2 x \Delta n_x
\]

\( C_2 \) = inertial resistance coefficient, \( v \) = inlet velocity and \( \Delta n_x \) = porous medium thickness. This study used empirical equation to define \( C_2 \). Pressure drop through a bank of tube calculated with equation:

\[
\Delta p = Eu \frac{\rho v^2}{2} Z
\]
\[ v = \text{inter tube velocity}, \quad \text{Eu} = \text{Euler number}, \quad z = \text{number of row.} \]

C2 can be defined by comparing Eq. 2 and Eq. 3, and the result as follow:

\[
C_2 = \left( \frac{d}{a} \right)^2 x Eu x z \frac{\Delta n_x}{\Delta n_x} \tag{4}
\]

Equation 5 and 6 from reference 5 used to define Euler Number (Eu). 1\textsuperscript{st} and 2\textsuperscript{nd} module of steam coil air preheater use equation 5 since it installed in inline configuration. 3\textsuperscript{rd} and 4\textsuperscript{th} module installed with staggered configuration and use equation 6.

\[
Eu = 0.52 \left( \frac{d}{d_e} \right)^{0.3} \left( \frac{n-1}{a-1} \right)^{0.68} Re_d^{-0.08} C_z \tag{5}
\]

\[
Eu = 2 \frac{\Delta p}{\rho x u^2} = 5.4 \left( \frac{d}{d_e} \right)^{0.3} Re_d^{-0.25} C_z \tag{6}
\]

C2 value for 1\textsuperscript{st}, 2\textsuperscript{nd} and 3\textsuperscript{rd}, 4\textsuperscript{th} module/tube bank are 15.2 and 35.9 respectively.

2.3 Validation

To validate the model, the study compares 3 parameters (outlet temperature, inlet pressure and pressure drop) between commissioning/calculation data and simulation result. Table 1 shown the validation of the model.

| Parameter          | Simulation Result | Reference Data | Error % |
|--------------------|-------------------|----------------|---------|
| Outlet Temp (°C)   | 82.55             | 83             | 0.54    |
| Inlet Pressure (Pa) | 6668.92           | 6668          | 0.001   |
| Pressure Drop (Pa) | 273.48            | 265.44        | 3.03    |

Table 1 above show simulation data closed with reference commissioning data. And to ensure that the model is not affected by the grid number of the mesh the study conduct 4 mesh simulation they are: 800,000, 1,220,000, 1,410,000 and 1,607,958. the result show maximum error for the grid is 0.019%.

2.4 Post Processing

To analyse the simulation result some qualitative and quantitative data are taken. They are temperature, pressure and velocity contour, graphic and facet average.

3. Result and Discussion

The simulation result in various data related to temperature, pressure and velocity along the flow in the Steam Coil Air Preheater. The data show the characteristic of a fluid flow through a set of tube bank heat exchanger, temperature rise, total pressure drop and velocity variation due to tube resistance and geometrical shape change.

Figure 3 and Figure 4 show the velocity and static pressure change along the flow for various position. The velocity of the flow will decrease with load decrease, this is as expected that lower load will be supplied with lower air amount. Even though that the value of the velocities and static pressure are different for each load, it can be observed that the velocity and static pressure change along the flow for a certain position have similar pattern for all load variation. The inlet is in position x = 6.5 m and outlet in position x = 0, the steam coil air preheater tube installed between position x = 4 and x =
3. From the velocity and static figure, it was observed that the velocity is high at the inlet since it has the smallest cross-sectional area. The velocity decreases and static pressure increases gradually when entering the transition duct that enlarges and its cross-sectional area increases gradually. The velocity and static pressure decrease when the flow enters the steam coil air preheater tube bank, as shown in all load variations. This is caused by the tube bank/module that restricts the flow and results in flow energy losses, which can also be observed from the total static pressure decrease shown in Figure 5. Figure 5 also shows that most of the total pressure decrease occurred at the steam coil air preheater tube bank, with the other parts showing only a small decrease in total pressure caused by friction with the duct wall. The velocity and pressure observations showed that the model can accurately figure out the characteristic of the flow through the steam coil air preheater.

**Figure 3.** Velocity profile along the flow with load variation

Figure 6 showed the stream trace and velocity contour from side and top views. There is a geometrical shape change of the duct, after the damper, the duct enlarges vertically and horizontally towards the Steam Coil Air Preheater module. After the module, the duct cross-sectional area gradually reduces to the outlet section. It was observed that the sudden geometrical shape change results in secondary flow at some area. Figure 6 Side view show that geometrical shape change is not symmetric, lower side have...
sharper change compare to upper side. Its also observed that there is offset between inlet ducts center line and outlet ducts center line. Those two factors make the secondary flow bigger in the lower side of the duct. Top view observation shows secondary flow at some area. Because it a symmetric geometrical shape change, it can be observed right and left side have similar secondary flow.

Figure 7 showed the temperature profile along the flow in the steam coil air preheater for various load. It observed that there is significant temperature rising between position x =4 m and position x = 3 m where the steam coil air preheater modelled as porous medium (PM) Installed. The temperature rising average are 40.79 °C, 31 °C and 29.34 °C for 100%, 75% and 50% load respectively. The temperature rising have a sharper gradient between position x = 3.5 m and position x = 4 m, this is because the 3rd and 4th steam coil air preheater module each has 2 rows of tube and 1st and 2nd module only has 1 row each. For all load variation also observed that liney1 and liney2 have bigger temperature rising compare to linez1 and linez2. This phenomenon occurs because line y1 and liney2 are in the lower half of the duct. There are bigger secondary flows on the lower half of the duct because asymmetric shape change (side view) that make the temperature slightly increase right before the flow enter the tube bank. And also because of this asymmetric geometrical shape change, the lower
half flow has longer flow path in the tube bank/module then the flow gets more heat transferred from the tube. All the phenomena above show that the porous medium model can simulate the heat exchanger in the CFD Software.

**Figure 5.** Total Pressure profile along the flow with load variation
Figure 6. Velocity contour and Streamtrace; Sideview (A), Top View (B)
The result data above showed that porous medium model with empirical equation to determine inertial resistance coefficient (C2) can simulate flow and heat transfer in steam coil air preheater tube bank. The validation data show the result closed to commissioning data. The flow characteristic mainly affected by geometrical shape and tube bank arrangement. Sudden change in cross sectional area affect velocity, pressure and temperature distribution. It also results secondary flows that contribute to flow energy losses.

4. Conclusion
This study show porous medium modelling can describe the characteristic heat and fluid flow in the steam coil air preheater heat exchanger. The alternative method to define inertial resistance coefficient (C2) is used. C2 define using empirical equation. This method can be an alternative to model heat exchanger using porous medium and not enough heat exchanger performance data to define inertial resistance coefficient. The simulation show the model result data close to commissioning data. It observed that the velocity and pressure distribution in the steam coil air heater affected by geometrical shape change and tube bank flow restriction. Sudden change in cross sectional area result in secondary flow. Install flow guide plate or modify the duct with smooth change in cross sectional area will reduce secondary flow. Temperature distribution affected by the flow characteristic (velocity, flow-path) and tube bank arrangement.

References
[1] C. Abeykoon, Compact heat exchanger – Design and optimization with CFD, International Journal of Heat and Mass Transfer 146 (2020) 118766
[2] M.M.A. Bhuta, N. Hayat, M.H. Bahir, A.R. Khan, K.N. Ahmad, S. Khan, CFD Application in Various Heat Exchanger Design: A Review, Applied Thermal Engineering 32 (2012),1 -12
[3] M. Musto, N. Bianco, G. Rotondo, F. Toscano, A Simplified Methodology to Simulate Heat Exchanger in an Aircraft’s Oil Cooler by means of Porous Media Model, Applied Thermal Engineering 94 (2016), 836 – 845

[4] W. Wang, J. Guo, S. Zhang, J. Yang, X. Ding, X. Zhan, Numerical study on hydrodynamic characteristics of plate fin heat exchanger using porous medium approach, Computer and Chemical Engineering 61 (2014) 30 -37.

[5] Yudin, V F., Tokhtarova, L. S. Lokshin, V., and Tulin, S. N., Correlation of Experimental Data on Convective Heat Transfer in Cross Flow Over Bundles with Transverse Spiral and Circumferential Fins, Trudy TsKTI, No. 82, 1968

[6] The Babcock & Wilcock Company (2005), STEAM its Generation and Use edition 41 (2005), The Babcock & Wilcock Company, Ohio.

[7] D. Brian Spalding, J. Taborek (1983), Heat Exchanger Design Handbook, Hemisphere Publishing Corporation, Washington.