Aerodynamic Study of Air to Gas Leakage Reduction in a Typical Rotary Regenerative Air Preheater of Coal-fired Steam Generators

Lidia Ferravante[1], Claudio Zagano[1], Valerio Marra[3]
[1]RSE S.p.A. - Ricerca sul Sistema Energetico - Via Rubattino 54 - 20134 Milano - MI- Italy
[3]COMSOL S.r.l. - Via Vittorio Emanuele II, 22 - 25122 Brescia - BS - Italy

Abstract: The present study relates to the reduction of the amount of air to gas leakage in a typical rotary regenerative air preheater of coal-fired steam generators, by means of computational thermo-fluid-dynamics. Due to the gaps, or clearances, required for rotation, there is a significant amount of leakage of the higher pressure air to the lower pressure gas stream. This paper presents a turbulent 2D CFD model, taking into account heat transfer also, of the preheater shown in Figure 1. Modeling and simulations have been carried out using COMSOL Multiphysics®. In order to reduce the driving force that pushes the air through the seals, flow patterns close to the seals were investigated in order to find an appropriate deflector (Figure 2). Different solutions have been studied; the one with the deflector has shown, in comparison to the original system, an effective leakage reduction of about 26%.
Keywords: Air Preheater, CFD, Fluid Flow, Heat Transfer

1. Introduction

The present study relates to the performances of coal-fired steam generators where we focused our attention to the reduction of the amount of air to gas leakage in a rotary regenerative air preheater.
On many boilers the phenomenon of leakage drift in the air preheater has had an adverse effect on performance. The most serious effect has been the loss of MW sent out from the station.
Due to the gaps, or clearances, required for rotation there is a significant amount of leakage of the higher pressure air to the lower pressure gas stream. After a period of operation up to 2 or 3 years the leakage level will increase, typically from 5% - 2% in the first days of run to between 12% - 25% (and often higher) on high pressure mill preheaters. After a period of 5 years the leakage can reach up to 35% - 45%.
Many types of seals and devices to adjust the gap size have been employed to reduce leakage. All these solutions suffer a progressive deterioration after some time. In this study, a non usual way to reduce the leakage has been investigated. It consists in an aerodynamics solution.

Figure 1: Typical rotary regenerative Air Preheater in phase of implementation in a coal-fired steam generators (Image courtesy of ALSTOM)

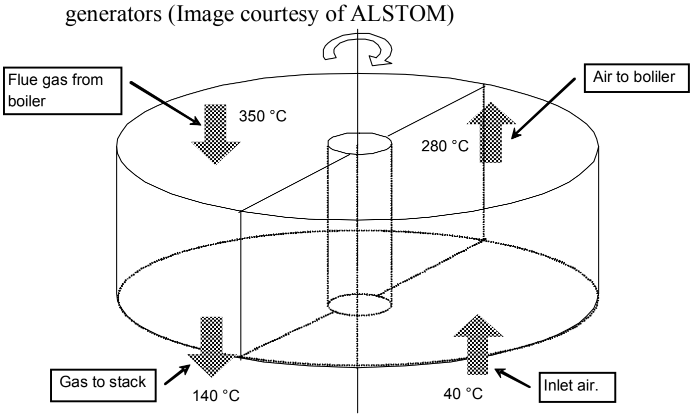

Figure 2: Simplified representation of the rotary regenerative air preheater (reference for the 2D model).

2. Modeling

The rotary regenerative air preheater is a rather complex machine. This has forced us to face the not easy task of simplifying it in a 2D model still representative of the real machine.
The 2D model of a rotary regenerative air preheater has been realized under the following assumptions:

- the simplified bidimensional geometry has been realized slicing the real 3D geometry thus considering the rotating effects during the normal duty of the application negligible.

- in order to correctly represent the real application, a 1:1 scale has been adopted in the drawing of the 2D model. The availability of structural and operative field data of the preheater has allowed checking a real-world modelling of the application.

- cause the very slow rotational velocity of the rotor (1 or 2 rotations per minute) in comparison to the main gas velocity, the rotor has been considered not rotating. So there are no moving parts in the model .

- the heat exchange elements have not been represented in the model. Thousand of hot end elements (undulate profile metal sheets normally made of carbon steel with thickness from 0.5 to 0.7 mm) are present in the rotor, so has been decided to design the basket frames without such elements and substituting them by means of a “volume force” reproducing the pressure drop due to them when crossed through by gases. The thermal conduction is ensured by setting an adequate value of Q (heat source) in the relevant subdomain setting.

The 2D model (Figure 3) consists in two parallel sections (A, B air; C, D gas). Each of them contains in the middle a part representing the rotor with the basket frames, which in the 2D design are modelled as parallel bars (E, F). Two very narrow sections (H, G) connecting the two big ones are positioned just over and under the rotor and represent the leaks of the sector plates (not visible) which must guarantee the seals.

The authors assume reasonable that the pressure variations involved are not affecting by a significant magnitude the density variations of the fluid. The density of the fluid is assumed to vary only because of the wide range of temperature variations.

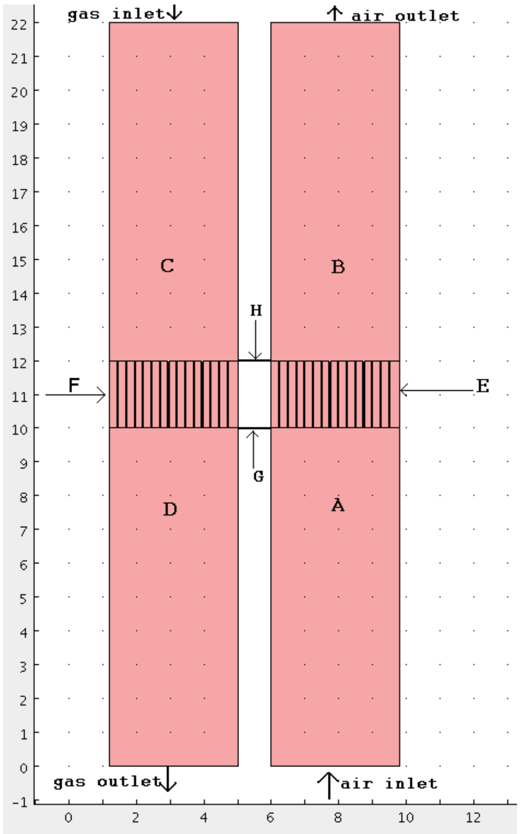

Figure 3: Simplified 2D model in real scale (m) of the rotary regenerative Air Preheater build in COMSOL.

The high velocity reached by the fluid (air) simulating the air leakage in the narrow sections represents a critical aspect for the model. The maximum velocities are around 100 – 110 m/s. Such velocities are next to 0.3 Mach (1 Mach = 340 m/s sound velocity in the atmosphere) that is the limit among the physics of the incompressible and compressible fluids. It has not been possible therefore to formulate the hypothesis of complete incompressibility so the relevant settings for a non isothermal and weakly compressible flow modelling have been chosen.
This modelling approach, velocities and pressure boundary conditions, the geometry of the wide sections (with a diameter of about 4 m), and the aspect ratio of the narrow ones (0.03 m of diameter) resulted in high turbulence regions within the geometry.

It has been necessary to use a turbulence model and tune several parameters in the physics of the weakly compressible turbulent flow in order to reach converge towards acceptable solutions.

Many aerodynamic profiles of deflectors have been studied with the aim to reduce the pressure near the inferior seal. After several simulations, a few profiles with high performances have been found.

2.1 Physics

The performance of a heat exchanger system where the flow is turbulent and with a Reynolds number around 10^6 has been investigated.

A coupled not predefined thermo-fluid-dynamic solution with non-isothermal turbulent flow with variable density has been chosen.
Values for inlet velocity, inlet temperature, heat flux, and outlet pressure obtained with this modeling approach results in air temperature and velocity that are consistent with real data..

2.1.1 Application Modes and Domain Equations

"Momentum Transport" – Turbulent Flow – k-ε Turbulence Model including effects of variable density (weak compressibility) in stationary mode analysis has been selected from the "Chemical Engineering Module" .
A Realizability Constraint has been applied in order to constrain the turbulent viscosity and avoid an unphysical production of turbulence at stagnation points.

As far as concerns the Heat Transfer from "Chemical Engineering Module" Energy Transport- Steady State Analysis- Convection and Conduction has been selected .

Air as material from the Material Library (since its properties are similar to the used gas) has been set in all subdomains.
In this way all material properties depending on Pressure and Temperature, as Density (ρ), Dynamic Viscosity (η), Thermal Conductivity (k), Heat Capacity (C_p) , are calculated automatically.

Regarding Fluid-Dynamics in subdomains E and F, representing rotor, volume forces have been introduced to simulate resistance that air and gas find passing through narrow air section passages among elements (specially formed sheets of heat transfer surface packed into the rotor).

The problem of how to define the "volume force" F that simulates the thermal exchange elements has been faced also.
It has been chosen to assimilate them to a porous media. The Forchheimer equation is commonly used for describe the pressure drop in the porous media and it is generally writing in the following way:

$$\frac{\Delta p}{L} = \alpha_1 u + \alpha_2 u^2$$

At the left side the term represents the pressure drop p for unit of crossed length L from the fluid in the porous media. To the right the first term represents the Blake-Kozeny term for laminar flow where the pressure drop is linearly dependent with the linear velocity u as predicted by Darcy low. To the right the second term represents the Burke-Plummer term for the turbulent flow where the pressure drop is proportional to the square of the velocity. The α_1 and α_2 are function of the porosity, viscosity, density of the fluid, and pores mean diameter. Forchheimer formulated for the α coefficients the following form:

$$\frac{\rho \varepsilon_p C_f}{\sqrt{k}} \mathbf{u} |\mathbf{u}|$$

where ρ is the density of the fluid, ε_p the porosity degree (dimensionless), k (m^2) represents the permeability of the porous media, while the coefficient of friction C_f is:

$$C_f = \frac{1{,}75}{\sqrt{150 \varepsilon_p^3}}$$

The terms defined as "volume force" F assume the following forms:

$$F_x = -\frac{\rho \varepsilon_p C}{\sqrt{k}} u \sqrt{u^2 + v^2}$$

$$F_y = -\frac{\rho \varepsilon_p C}{\sqrt{k}} v \sqrt{u^2 + v^2}$$

In our case only the component in the direction of the flow (Fy) has been considered, since all

elements are parallel to the y direction and cannot be crossed.
These forces Fy have been calculated in a proper way to achieve a correct pressure-velocity correlation for inlet and outlet in both air and gas streams.

In order to improve convergence when solving the model, it's common use to introduce artificial diffusion. This means that the spurious oscillations can be removed without refining the mesh till the resolution beyond which the discretization is stable.
Navier-Stokes equations have been stabilized by streamline, crosswind, and isotropic diffusion.
Turbulence equations have been stabilized by isotropic and streamline diffusion as some extra diffusion is necessary to capture the shear layers and prevent the computation from diverging.

As density is a function of the temperature, the momentum equations, continuity equation and temperature equation form a closely coupled system of equations. For optimal performance, this coupling must be accounted for in the stabilization expressions. The model has been set up in such a way providing the fluid dynamic with the temperature variable and the heat transfer with the velocity field. Moreover, all material properties are temperature dependent. .

Heat Transfer in subdomains E and F, representing rotor, has been defined also in terms of a uniform heat sources Q that describes heat generation within the domain, express heating and cooling with positive and negative values, respectively in order to simulate the proper heat exchange among elements and air. They have been set so to give correct values of temperature at air and gas outlets starting from fixed inlets temperatures.
k_T, turbulent thermal conductivity, has been set manually (k_T=C_chcc*etaT_chns) so to couple heat transfer with fluid-dynamics by means of turbulent dynamic viscosity computed by the turbulence model.

Stabilization is important in Heat Transfer too. It is needed because pure Galerkin discretization (the finite element discretization method underlying convection-diffusion transport equation in COMSOL Multiphysics.) is unstable for convection-dominated or source-term dominated transport equations. So streamline, crosswind and isotropic diffusion have been activated.

2.1.2 Boundary Conditions

Boundary conditions in the fluid domain are:

- specified velocity at the inlets (air and gas entrance) with adequate turbulent length scale L_T to take in account boundary layer effect. The turbulent length scale L_T is a measure of the size of the eddies that are not resolved, for pipes and channels it is suggested to set L to 0.07*L, where L is pipe/channel diameter.
- specified pressure at the outlets (air and gas boundaries).
- logarithmic wall function at all other boundaries.

Boundary conditions for the heat exchange are:

- specified temperature at the inlets (air and gas entrance).
- convection-dominated transport at the outlets (air and gas outlets).
- thermal wall function at all other boundaries.

The cold air stream flow rate in the air inlet duct (subdomain A, Figure 3) is 7.8 m/s at a temperature of 40 °C. The temperature of the air after rotor (subdomain B) is about 290 °C and has a flow rate of 11.4 m/s.
The hot gas stream flow rate in the gas inlet duct (subdomain C) is 15.3 m/s at a temperature of 350 °C.
The temperature of gas after heat exchange (subdomain D) is about 150.°C and has a flow rate of 11 m/s.

2.1.3 Mesh and Solver

To solve the problem and get an accurate solution, the mesh must be fine where there are sudden changes in direction of the flow and where there are narrow passages.
To generate such a mesh has been used the method of triangle advancing front.
The base mesh used in the model consists approximately 2,848,500 degrees of freedom (DOFs).
In Figure 4 a particular of mesh in critical points is shown.

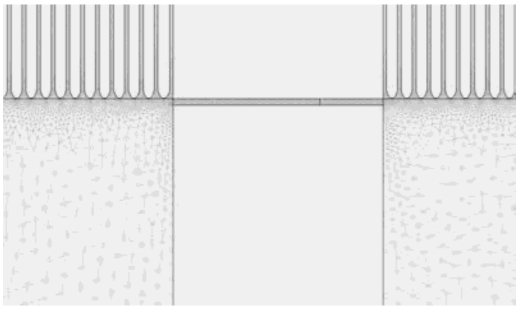

Figure 4: details of the mesh.

The Navier-Stokes equations and the turbulence equations form an equation system is very hard to solve froma numerical point of view in a coupled manner, therefore is common practice to segregate the equations system.
To solve the equations system, the stationary segregated solver has been used where the equations system is separated into three groups:

- the first group contains u, v, and p as variables (x direction velocity, y direction velocity, pressure).
- the second group contains the turbulence variables logd and logk.
- the third group contains the variable T (temperature).

To help the convergence a manual scaling with reasonable values of variables has been done. Reasonable values of variables have been found with some preliminary rough iterations.

3 Results

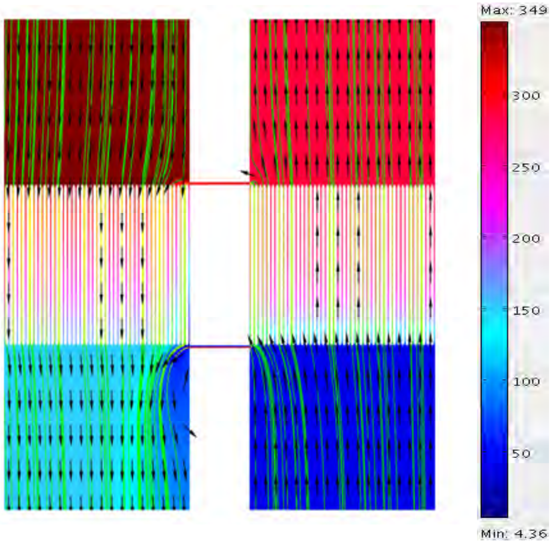

Figure 5: temperature distribution (°C) in the Air Preheater 2D model.

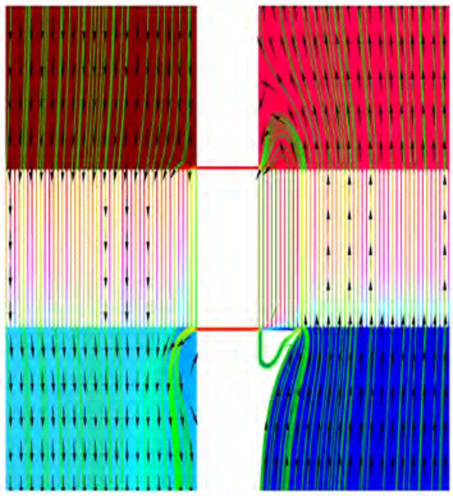

Figure 6: temperature distribution (°C) in the Air Preheater 2D model with optimized flux deflectors.

The COSMOL Multiphysics 2D model of the Air Preheater has given fitting with real data results.

To reduce of the amount of air to gas leakage many aerodynamic profiles of deflectors have been studied.

Deflectors have been designed to reduce the pressure near the inferior seal.

After several simulations, a few profiles with high performances have been chosen. From the quantitative point of view, these deflectors reduce the leakage of air to about 15 kg/s, (22,2 kg/s is the value without deflector).

Acknowledgements:

This work has been financed by the Research Fund for the Italian Electrical System under the Contract Agreement between ERSE (now RSE – Ricerca sul Sistema Energetico) and the Ministry of Economic Development - General Directorate for Nuclear Energy, Renewable Energy and Energy Efficiency stipulated on July 29, 2009 in compliance with the Decree of March 19, 2009.