Strojniški vestnik - Journal of Mechanical Engineering 51(2005)7-8, 411-417
UDK - UDC 536.2
Izvirni znanstveni članek - Original scientific paper (1.01)

Numerical analysis of heat transfer and fluid flow in rotary regenerative air pre-heaters

Sandira Alagić[1], Nikola Stošić[2], Ahmed Kovačević[2], Indira Buljubašić[1]

[1]*Faculty of Mechanical Engineering, University of Tuzla, Univerzitetska 4, Tuzla, BiH, sandira.alagic@untz.ba*
[2] *School of Engineering and Mathematical Sciences, City University London, Northampton Square, EC1V 0HB, UK*

Abstract

The Ljungström rotary air pre-heater is a regenerative heat exchanger used for preheating the combustion air, mainly in steam boiler plant. The hot gas and cold air ducts are arranged to allow both the flue gas and the inlet air to flow simultaneously through the machine. The hot flue gas heats the rotor material and as the rotor rotates, the hot rotor section moves into the flow of the cold air and preheats it.

Existing simulations of regenerative air pre-heaters are mainly based on empirical approximations where some of the effects of the process are neglected. Although this usually gives reasonably acceptable results, it was thought that CCM analysis of such devices, which stands for Computational Continuum Mechanics, would result in a better understanding of the process features, such as the fluid-solid interaction.

A grid interface was developed to transfer the geometry of such a Ljungström air pre-heater to a finite volume numerical mesh which is later used for the calculation of unsteady fluid-solid interactions. Results were obtained by use of the commercial CCM solver ”Comet” of Star CD. In this paper the results are presented in the form of diagrams of the velocity and temperature fields as functions of time and space. The results of both the one- and three-dimensional calculations and field measurements are compared and good agreement was achieved.

The result of this study is the development of an effective procedure for computer calculation of processes in a Ljungström air pre-heater to optimise its parameters, which can be used either for research and development or in everyday engineering practice.

Introduction

Much effort has been spent to maximise the efficiency of each stage of the transformation of the chemical energy of fuel to electrical power. In a steam boiler, the most significant loss is associated with the energy of the outgoing flue gas. This can be reduced by the use of an air pre-heater which transfers energy from the outgoing exhaust gases, to the incoming air, prior to combustion. The Ljungström air pre-heater, as shown in Figure 1, is a regenerative heat exchanger often used for this function in industrial power plant. In it, the hot gas and cold air are arranged to flow in opposite directions through parallel ducts, each of which, passes through a section of a rotor. The combustion products heat the rotor as it revolves through that section. Further rotation of the rotor brings it into contact with the incoming air, where it is cooled by it. The air is thereby preheated before passing to the boiler furnace, where it is used for combustion of the fuel. The rotor is divided into a number of sections, which are separated by seals, in order to prevent mixing of the flue gases and combustion air. These sections consist of small passages formed by profiled sheets, as shown in Figure 2, and the performance of the air pre-heater depends on the size and shape of these small cells.

A number of studies have already been published, which describe methods of estimating the performance of regenerative pre-heaters. However, all of them are based on dimensionless or one-dimensional models in which some significant effects are either neglected or estimated empirically. A three dimensional approach was therefore regarded as a useful tool to analyse heat and fluid flow within the cell elements more precisely and how varying their size and shape would affect the interaction between the rotor and the fluid.

Nomenclature

A	- area of contact between the solid body and gas, area	c	- specific heat
e	- internal energy	C	- turbulence model constants
h	- heat transfer coefficient, enthalpy	**f**	- body force
I	- unit tensor	**i**	- unit vector
m	- mass	k	- conductivity, kinetic energy of turbulence
P	- production of kinetic energy of turbulence	p	- pressure
s	- control volume surface	**q**	- source term
u	- displacement in solid	t	- time
V	- volume	**v**	- fluid velocity
z	- axial coordinate	x	- spatial coordinate
α	- temperature dilatation coefficient	Γ	- diffusion coefficient
ε	- dissipation of kinetic energy of turbulence	ϕ	- variable
λ	- Lame coefficient	μ	- viscosity
η	- Lame coefficient	ρ	- density
σ	- Prandtl number		

Indices

eff	- effective	g	- gas
in	- inflow	out	- outflow
s	- solid	T	- turbulent

The use of Computational Fluid Dynamics, CFD, to analyse the flow in a variety of thermal equipment has been widely reported, but there is no record in the open literature of its use and use of CCM for calculations of flows in Ljungström air pre-heaters. One possible reason for this is the complexity of the geometry and flow within these devices.

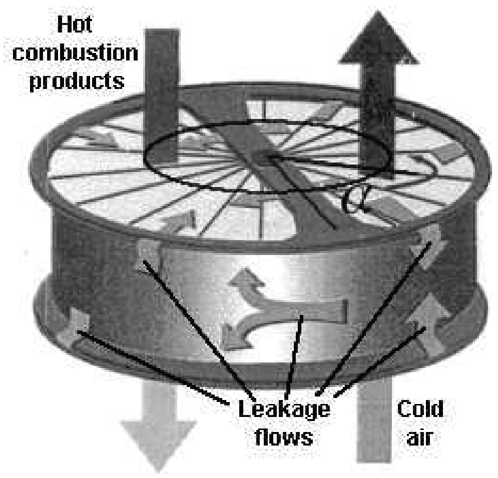

Figure 1 Fluid flows in rotary regenerator

A further complication of the analysis is that the flow processes involved are unsteady and their nature can only be properly understood by consideration of the interaction between the fluid flow and solid structure.

More recently, the use of continuum mechanics and a substantial increase in computer speed and capacity have made it possible to develop specialized computer codes for simultaneous application of finite volume numerical methods to both fluid flow and its surrounding solid structure in the form of commercial CCM codes.

In order to apply these advanced numerical methods to the calculation of rotary regenerator devices, the authors have developed a general and flexible grid generation procedure. An interface program written in FORTRAN enables an arbitrary geometry of a pre-heater to be automatically mapped with the discrete volumes and then used for calculation in a standard CCM code. By this means, a study was carried out on an existing pre-heater in "Tuzla" power station, using the commercial CCM solver "Comet". The results are displayed in the form of the temperature distribution within the pre-heater solid elements and fluid flow of both the hot combustion products and cold air as a function of both time and spatial position

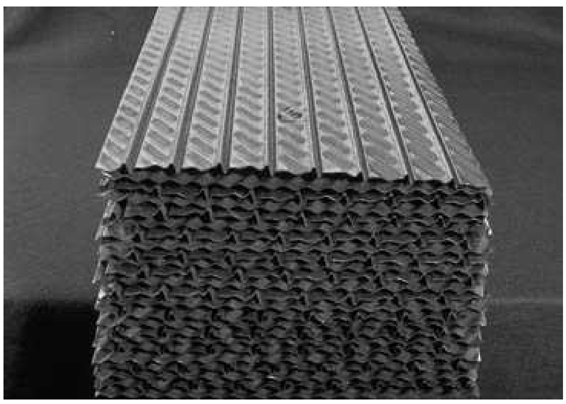

Figure 2 Filling of a rotary regenerator

One-dimensional energy balance of the pre-heater

A one-dimensional model is developed and described in this paper, which assumes that each control volume consists of solid and fluid parts connected via a common area surface. The solid and fluid components of the cell exchange heat through certain regions of the known surface area. As a result of the rotation of the device, the temperature within it changes continuously both in space and time.

In such a case, a one-dimensional numerical model of heat transfer between the hot combustion products and solid material on one side, and the cold air and solid body on another, can be applied to estimate the pre-heater performance reasonably well. [1]

The control volumes are positioned next to each other and a constant speed of rotation is assumed for all of them.

The mathematical model utilized here consists of two energy balance equations, one for the solid body, (1) and one for the fluid phase, (2). The interface between these two equations is the convective heat transfer between the fluid and the solid.

$$\underbrace{m_s c_s \frac{\partial T_s}{\partial t}}_{\substack{accumulation\\ in\ solid\ body}} + \underbrace{k_s A_i \frac{\partial^2 T_s}{\partial x_i^{\,2}}}_{\substack{conduction\\ in\ solid}} = \underbrace{hA\left(T_g - T_s\right)}_{heat\ source} \tag{1}$$

$$\underbrace{m_g c_p \frac{\partial T_g}{\partial t}}_{\substack{accumulation\\ in\ gas}} + \underbrace{\dot{m}_g c_p \left(T_{in} - T_{out}\right)}_{convection} + \underbrace{k_g A_i \frac{\partial^2 T_g}{\partial x_i^{\,2}}}_{conduction\ in\ gas} = \underbrace{hA\left(T_g - T_s\right)}_{heat\ source} \tag{2}$$

$$m_s c_s \frac{\partial T_s}{\partial t} + k_s A_z \frac{\partial^2 T_s}{\partial z^2} = hA\left(T_g - T_s\right) \tag{3}$$

$$m_g c_p \frac{\partial T_g}{\partial t} + \dot{m}_g c_p \frac{\partial T_g}{\partial z} = hA\left(T_g - T_s\right) \tag{4}$$

Since each flow passage is narrow and surrounded by thin material, the fluid temperature change in the direction perpendicular to the main flow can be regarded as small and neglected. Also, since it is assumed that heat is not exchanged with the surroundings in the radial direction, conduction in the solid body only occurs along the axial coordinate z. Heat transfer by conduction in the fluid is neglected because of the low fluid thermal conductivity. By means of these simplifications, the equations are reduced to those of unsteady 1-D flow, as shown in equations (3) and (4). These can be solved by a finite difference method.

Depending on the spatial position of the calculating domain, i.e. the angle of rotation, the fluid medium in these equations is either air or combustion products. The inlet temperatures and mass flow rates are assumed to be constant during the each half-period of the process and their assumed values are those obtained from industrial plant measurements. The spatial domain is discretised by assuming constant steps in the axial direction.

The time step is assumed to be constant. The resulting system of algebraic equations is then solved numerically by iterative procedure contained in the computer program written by the authors. The results obtained are in the form of instantaneous temperatures of the flue gas or air and solid, and the mass flows of the flue gas and air at every point along the axial coordinate z.

Three-dimensional analysis of the pre-heater

Both, the fluid flow and structural behaviour of the solid parts in a regenerative pre-heater are fully described by the mass averaged equations of continuity, momentum and energy conservation which are accompanied by equations of the turbulence model and state, as given, for example, in [4]. The solution of these equations is then made possible by inclusion of constitutive relations in the form of Stoke's and Fourier's law for the fluid momentum and energy equations respectively and Hooke's law for the momentum equations of the thermo-elastic solid body. The generic 3-D transport equation is then given as:

$$\frac{d}{dt}\int_V \rho\phi dV + \int_S \rho\phi \mathbf{v}\cdot d\mathbf{s} = \int_S \Gamma_\phi \operatorname{grad}\phi \cdot d\mathbf{s} + \int_S \mathbf{q}_{\phi S}\cdot d\mathbf{s} + \int_V \mathbf{q}_{\phi V}\cdot dV \tag{5}$$

The terms in the equation which describe the pre-heater case are given in *Table 1.*

The resulting system of partial differential equations is then discretised by means of a finite volume method in a general Cartesian coordinate system. This method maintains the conservation of the governing equations, while at the same time enables a coupled system of equations for both, solid and fluid parts to be solved simultaneously. Connection between the solid and fluid parts is explicitly determined if the temperature on the solid body surface is a boundary condition for the fluid flow and vice versa. The numerical grid, as explained in the next section, is attached to the CCM solver to obtain the distribution of the fluid temperature and velocity throughout the fluid domain and the temperature of the solid elements.

This mathematical scheme is accompanied by the boundary conditions for both the solid and fluid parts. Whether the fluid part contains hot gas or cold air it is entirely surrounded by the walls. Cyclic boundary conditions are applied to all sides of the domain except to the top and bottom parts. These are represented either as inlet or outlet. The initial values for all physical variables are given at the centre of each numerical cell within the domain and preset to the values of the air inlet flow.

Table 1 Terms in the generic transport equation (5)

Equation	ϕ	Γ_ϕ	$\mathbf{q}_{\phi S}$	$\mathbf{Q}_{\phi V}$
Fluid Continuity	1	0	0	0
Fluid Momentum	v_i	μ_{eff}	$\left[\mu_{eff}\left(\operatorname{grad}\mathbf{v}\right)^{\mathrm{T}} - \left(\frac{2}{3}\mu_{eff}\operatorname{div}\mathbf{v} + p\right)\mathbf{I}\right]\cdot\mathbf{i}_i$	$\mathrm{f}_{\mathrm{b},i}$
Solid Momentum	$\frac{\partial u_i}{\partial t}$	η	$\left[\eta\left(\operatorname{grad}\mathbf{u}\right)^{\mathrm{T}} + \left(\lambda\operatorname{div}\mathbf{u} - 3\alpha\Delta T\right)\mathbf{I}\right]\cdot\mathbf{i}_i$	$\mathrm{f}_{\mathrm{b},i}$
Energy	e	$\frac{k}{\partial e/\partial T} + \frac{\mu_t}{\sigma_T}$	$-\frac{k}{\partial e/\partial T}\frac{\partial e}{\partial p}\cdot\operatorname{grad}p$	$\mathbf{T}$: grad $\mathbf{v}$ + h
Turbulent kinetic energy	K	$\mu + \frac{\mu_t}{\sigma_k}$	0	$P - \rho\varepsilon$
Dissipation	ε	$\mu + \frac{\mu_t}{\sigma_\varepsilon}$	0	$C_1 P\frac{\varepsilon}{k} - C_2\rho\frac{\varepsilon^2}{k} - C_3\rho\varepsilon\operatorname{div}\mathbf{v}$

Grid generation

To solve the equations numerically, the spatial domain of the pre-heater has to be replaced by a numerical grid that contains discrete volumes. This process of replacing the spatial domain by a system of grid points is called numerical grid generation. Both, the type and the quality of a numerical grid play important role in the accuracy of the numerical solution and efficiency of the finite volume method [4].

A composite grid, made of several structured grid blocks patched together and based on a single boundary fitted co-ordinate system is used to transform the physical geometry of the selected part of an air pre-heater into discrete volumes. Grid blocks are then connected over the defined regions through their boundaries, which coincide with other elements of the entire numerical mesh. Boundary fitted and confirmed numerical meshes are generated for each part of both, the fluid and solid parts as in Figure 3, of ref [6].

In order to make the 3-D simulation and evaluation of the air pre-heater process faster and easier, only one cell of the accumulation mass in the radial and circumferential direction was considered for calculation.

The grid calculation method applied in this paper is based on algebraic transfinite interpolation with multi parameter adaptation of the boundaries. This includes stretching functions in order to ensure both grid orthogonality and smoothness.

More information about analytical grid generation methods can be found in refs [5] and [6].

The grid generation method is implemented in a pre-processor program developed by the authors in order to produce a numerical mesh suitable for analysis of a Ljungström rotary heat regenerator and to incorporate it into existing finite volume CFD and CCM software automatically.

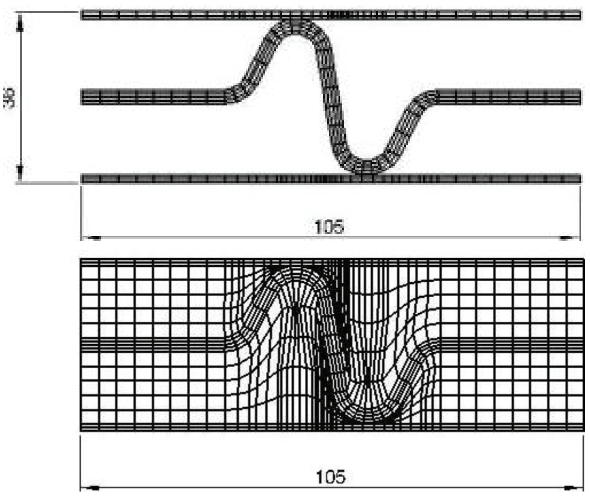

Figure 3 Cross section through the numerical mesh: Top – fluid part, Bottom - solid and fluid part

Comparison of 1-D and 3-D simulations

The analysis of processes in a rotary air pre-heater was performed for steam boiler No. V, Unit IV in Tuzla Power Plant.

A numerical solution was obtained for the working conditions specified in the operational documents [7]. These are: speed of rotation 1.76 rpm, air inlet temperature 313 K, gas inlet temperature 588 K, air inlet velocity 4.5 m/s and gas inlet velocity -8.0 m/s. The negative value means that gas enters the computational domain in the opposite direction to the air, as shown in Figure 1. Dimensions of the regenerator are: outer radius 3.8 m, inner radius 0.5 m, height 1.3 m.

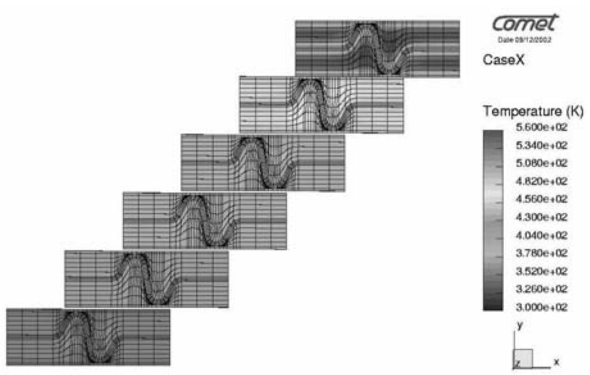

Figure 4 Time step 1, Rotor angle 3.36

Cold air enters the computational domain when the computational angles of rotation are between 0° and 150°. That is the angle range for which the pre-heater filling is exposed to the air channel. Similarly, the combustion products are in contact with the computation domain between 180-350°.

For other values of the rotational angle, both sides of the computational domain are closed and the velocities at their boundaries set at 0.

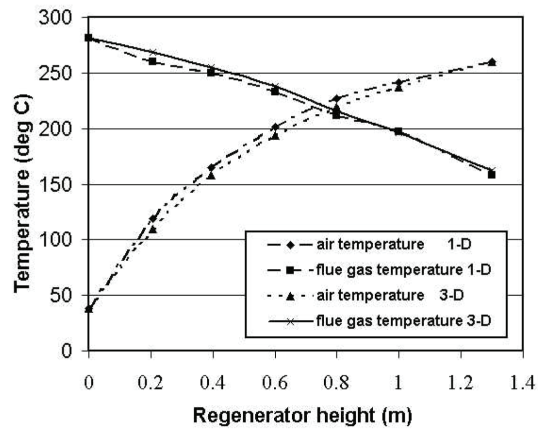

Figure 5 Comparison of temperatures calculated with 1-D and 3-D models

Figure 4 shows results of the 3-D analysis. The first time step, which is presented, corresponds to 3.36°. This represents the position at which the control volume becomes exposed to the cold air stream. That part of the process ends at 150°. In that period, heat is transferred from the solid part of the domain to the cold air, which, results in its temperature rise from 38°C to about 263°C.

During the second period, which finishes at the angle of 180°, the gas contained in the computational domain tends to an average temperature along the height of the pre-heater. However, that time period is too short for such a process and the maximum gas temperature remains at the level of 260°C. The computational domain then becomes open to the hot combustion products at 180°. During the following period, most of the heat is transferred to the cold metal sheets, which are heated to the highest temperature difference while the hot gas is cooled to about 177°C in the 62nd time step. A comparison between the results of the calculated air and gas temperatures obtained by both the 1-D and 3-D models is given in Figure 5. The temperature distribution is given along the complete height of the regenerator. Good agreement between the two models is achieved.

A comparison of both, the fluid flow and solid body temperatures at different heights as a function of the angle of rotation is shown in Figure 6. In the same figure, the variation in the temperature of the air, heated due to cooling of the metal sheets, as well as change in the temperature of the flue gas, which is cooled due to heating of the metal sheets, is shown. It can be concluded from these two diagrams that the 3-D results give good agreement with the corresponding results obtained by the 1-D model.

The results of this comparison between the 1-D and 3-D numerical results and experimental data confirm that the 1-D model is adequate for most purposes when using CCM procedures to estimate pre-heater performance, but the 3-D model increases a level of confidence of using numerical results.

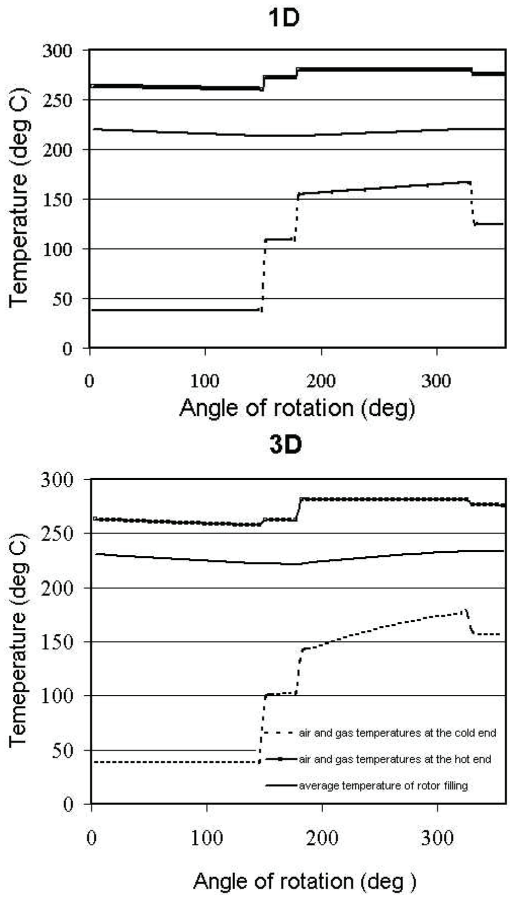

Figure 6 Temperatures of fluid and solid part obtained by 1-D and 3-D calculations

Experimental investigation and analysis

A further stage in the investigation was to determine how reliable the 1-D and 3-D models were in predicting real plant performance when compared with results obtained from power plant measurements.

Tests were therefore carried out on the regenerative heat exchanger of the pre-heater 3, steam boiler V, Unit IV at Tuzla Power Plant to obtain temperature measurements at the boiler hot and cold ends. Values of the flue gas temperature were taken at seven measuring points. A total of 28 temperature measurements were taken with thermocouple probes at depths of 0.5, 1, 1.5 and 2m in order to get temperature fields across the pre-heater cross section at four different depths.

The gas temperatures measured at the exit of the air pre-heater for one operational point at a rotational speed of 1.76 rpm are given in Table 2.

The results of the 1-D and 3-D models, shown in Figure 6 show reasonable agreement with the measured values, as shown in Table 3.

The relative error between the 3-D calculations and measurements for the flue gas side was 5.82 % while for the air side the error was 0.03 %. Relative error for the 1-D calculations and measurements was 6.41 % for flue gas and 0.81 % for air.

Table 2 Measured temperatures in °C for air pre-heater 3, steam boiler V, Unit IV, Tuzla Power Plant

Height	Measurement points						
(m)	1	2	3	4	5	6	7
0.5	161	168	168	158	178	179	183
1.0	162	167	167	168	178	181	183
1.5	161	167	171	168	175	182	181
2.0	162	168	169	168	176	182	180

Table 3 Comparison between measured and calculated values for the regenerator speed n=1.76 rpm

Model	Temperatures of flue gas at cold end (°C)	Temperatures of air at the hot end (°C)	Relative error gas/air (%)
1-D	160.80	262.13	6.41/0.81
3-D	161.82	260.10	5.82/0.03
Measured	171.82	260.00	-

Speed optimisation

Since the heat exchanged within the air pre-heater depends on the rotor speed, a thorrough numerical study was performed to find out the temperature behaviour of the Ljungström pre-heater with speed variation.

The gas outlet and air inlet temperatures in function of the rotor speed are presented in Fig 7. As it may be noticed, at low speeds air exits with lower temperature, while the flue gases leave the pre-heater with higher temperature because both, air and gas are in contact with the pre-heater walls for longer period. Therefore, the heat exchanged is small. At zero speed, for example, there virtually will be no heat transfer at all and the exit temperatures will be the same as the inlet ones. At higher rotor speeds, air and gas temperatures will tend to a single value which balances depending on heat capacity of both streams. At infinite speed for example, the heat transferred will be the same as in a parallel flow heat exchanger of infinte heat transfer surface. However, since a well designed Ljungstroem pre-heater performs beter than a parallel flow heat exchanger, there must exist a rotor speed for which the heat exchanged reaches its maximum. This was estimated at approximately 3 rpm for the pre-heater in question. It appeared that 3.2 % of increase in heat transfer was obtained for that speed in comparison with the pre-heater operational speed of 1.76 rpm.

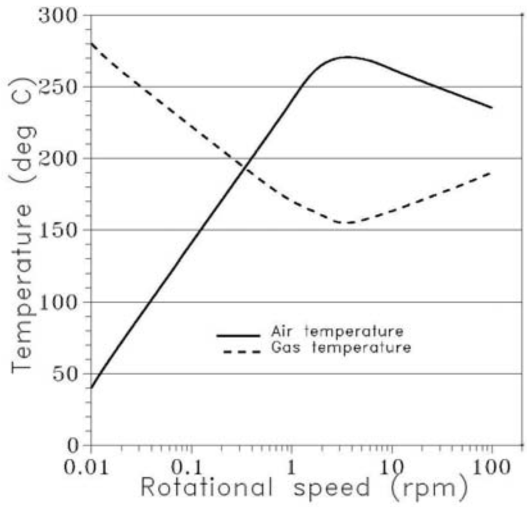

Figure 7 Exit air and gas temperatures in function of rotational speed

Conclusions

The aim of the investigation described in this paper was to obtain a better understanding of the processes in rotary air pre-heaters by the use of numerical estimation and experimental measurement of both the air and combustion product flows.

A convenient feature of rotary air pre-heater construction is that the entire machine can be analysed by consideration of only one gas and airflow path bounded by the filling metal material. The computational domain thus defined, can then be conveniently subdivided into a finite number of control volumes and analysed by the use of iterative procedures. The use of CCM, both in 1-D finite difference and 3-D finite volume methods, demonstrated that for most purposes, the 1-D model is adequate and demonstrated that this method both simplifies the problem statement significantly and speeds up the calculation procedure and that the 3-D model serves for better understanding of the pre-heater processes.

The experimental results confirm the validity of both the numerical analysis methods not only for calculation of the fluid flow temperatures but also for the temperatures of the pre-heater solid filling. By this means, a powerful tool was developed which allows a better understanding of the fluid flow and heat transfer process in rotary pre-heaters.

References

[1] E. Zafeiriou, D. Wurz: Numerical Simulation of Heat Transfer Processes in Rotating Regenerators, VGB Kraftwerkstechnik 76, Number 6, 1996.

[2] H. Klein, G. Eigenberger : Approximate solutions for metallic regenerative heat exchangers, International Journal of Heat and Mass Transfer 44, (3553 – 3563), 2001.

[3] B. S. Baclic, G. D. Dragutinovic: Operation of Counterflow Regenerators; Computational Mechanics Publications Southampton, UK and Boston, USA, 1998.

[4] J. H.Ferziger, M. Perić: Computational Methods for Fluid Dynamics, Springer, Berlin, 1996.

[5] J. F. Thompson, B. Soni; N. P. Weatherrill, “Handbook of Grid generation”, CRC Press 1999.

[6] A. Kovačević, N. Stošić and I. K. Smith: Three Dimensional Numerical Analysis of Screw Compressor Performance, Journal of Computational Methods in Sciences and Engineering, vol. 3, no. 2, 2003, pp. 259- 284

[7] Documentation of the Power Plant Tuzla, Unit IV