

Comparative Study of Experimental & CFD Analysis of Thermal Regenerator

Kuldeep Panwar, Satyendra Singh, Rishikesh Kumar

Abstract— The paper aims to study the comparative study of experimental and CFD analysis of packed-bed thermal regenerator. In the present work the previously published experimental study of regenerator with $D/d_p > 15$ are simulated and are compared with the simulated results of CFD. A very good agreement is seen between both the results in all flow regimes. Commercial Ansys Fluent software is used for the analysis.

Index Terms— CFD, unsteady, regenerator, thermal characteristic, Ansys Fluent

1 INTRODUCTION

Thermal Heat regenerators are the devices which are used for heat recovery or waste heat utilization. Regenerators are type of heat exchangers in which the heat transfer from one fluid to another is done by the help of intermediate solids. Before discussing the working principle of thermal heat regenerator firstly it is essential to study the classification of Heat exchangers. Heat exchangers are well known devices which are used to transfer heat from one fluid medium to another fluid. On the basis of how this heat is transferred between the fluids heat exchangers are broadly classified as Recuperators and Regenerators. Figure 1 (a) & (b) shows the simple difference between the method of heat transfer in recuperators and regenerator.

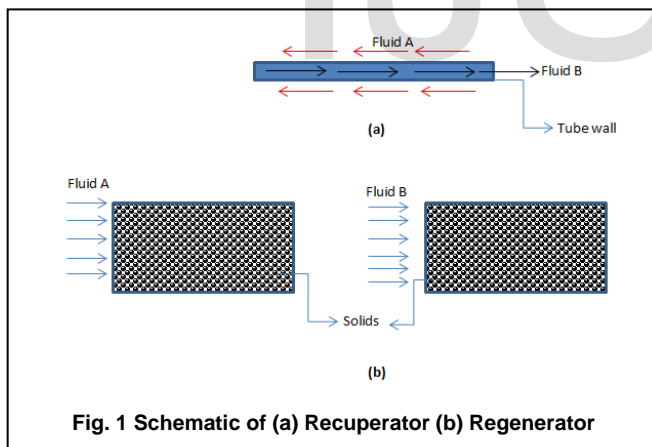


Fig. 1 Schematic of (a) Recuperator (b) Regenerator

In recuperators the heat transfer between two fluids is done through a wall which separates both the fluids as shown in Fig. 1 (a). On the other hand regenerators are the type of heat exchangers in which the heat between the two fluids is transferred with the help of intermediate medium

i.e. solids, solid having high volumetric capacity than gas are used to store the heat from the hot gas and transfer it to cold gas.

2 REVIEW OF LITERATURE

Dalman et al. [1] were one of the first researchers who used CFD to study the regenerator. They developed a two dimensional CFD model and solved it using finite element technique. A similar study with 8 spheres in a linear array inside a regenerator using FIDAP was done by Lloyd and Boehm [2].

The simulation helped to explain the flow and heat transfer through the regenerator bed. They conducted the simulation for a range of Reynolds number of 40, 80, 120 and Prandtl number ranging from 0.73 to 7.3.

Logtenberg and Dixon [3] with the help of CFD codes in FLOTTRAN modeled a regenerator bed with 8 spherical particles in two layers with particles perpendicular to the flow direction. The simulation was conducted for Reynolds number ranging 9 to 1450. Callis et al. [4] used CFX CFD software to simulate the pressure drop and drag coefficient in a square regenerator channel. The hydraulic diameter of regenerator to diameter of particle ratio (D/d_p) for the simulation was ranged from 1 to 2. The number particles were also increased upto 40 particles than the previous studies. The simulation done by them was validated with experimental results calculated used Laser Doppler Anemometer. The limitation of the study was its inability to explain wall effects in low D/d_p ratio. A quantitative comparison between experimental results and CFD predictions were presented by Michiel Nifemeisland and Dixon [5]. They simulated a regenerator model using CFD having 44 spherical particles. The D/d_p ratio was kept 2 to maintain the uniformity in void fraction. The simulation was focused to predict the velocity variation and temperature variation especially at wall-particle interface. The simulated predictions were compared with experimental setup of similar geometry.

Reddy and Joshi [6] presented CFD analysis of regenerator with column to particle diameter ratio as 5. The wall effect on the flow and heat transfer was studied in the paper.

Another CFD analysis of regenerator of a thermo caustic engine was done by David et al. [7]. They considered two

- Kuldeep Panwar is currently Assistant Professor in mechanical engineering in shivalik College of Engineering, India, PH-01352693401. E-mail: kuldeppanwar.kec@gmail.com
- Rishikesh Kumar is currently Assistant Professor in mechanical engineering in Skyline institute of Engineering & technology, India, PH-9457366553. E-mail: rishikesh.40058@gmail.com
- S. Singh is currently Professor in Mechanical Engineering in B.T.K.I.T. Dwarahat

modes of heat transfer mainly convection and radiation with in the regenerator. Ansys Fluent CFD tool was used for the analysis. The important findings of the paper included the effect of convection and its effects on amplitude and vibration of oscillating flow and effect of radiation on the performance of regenerator.

The recent research in the field of regenerator using CFD was presented by **Y. Lu et al. [8]**. A three dimensional, transient mathematical model was developed with spherical packing bed of aluminium. The model was solved by CFD FLUENT tool. The simulated results were validated with experimental results and the variation was found to be up to 4 %. The heating and cooling cycles were continued till the steady state was reached, which took 25 alternate cycles. The important findings of the paper were, with decrease in particle diameter the pressure drop increases due to flow resistance which in turn decreases the fluid flow velocity in the regenerator. In the present thesis the CFD simulation is first validated with the model of **Y. Lu et al. [8]** and when the validations comes in agreement with the experimental results of **Y. Lu et al. (2014)** then only it was assumed that the current CFD simulated regenerator model can be used for the further analysis.

The most recent research done by **K. Panwar and D. S. Murthy [9]** aims to provide a computational fluid dynamics (CFD) analysis of Pebble bed regenerator for transient flow powered by solar air heater. In their paper flue gas at 1000C enters at one end of the regenerator and the amount of heat absorbed by the pebbles was studied by the Ansys Fluent software. A regenerator physical model was developed to estimate actual performance of regenerator under transient flow condition. The proposed model was used to simulate the behavior of regenerator working under different operating and design condition. The standard k-e turbulent model with standard wall function is used for modeling of gas flow. The finding of the paper were that it was observed that pressure was gradually decreasing in the direction of gas flow, with the decrease in pressure in the regenerator velocity along the flow direction increased up to 9 m/s at the outlet of regenerator from 5 m/s at the inlet of regenerator. It was observed that the heat is taken by regenerator from hot air during heating cycle which could be seen by increase in temperature of regenerator bed. The temperature profile of regenerator bed showed the spreading of temperature front in regenerator due to the various factors such as deviation from plug flow of hot gas in porous medium resistance to heat transfer offered by fluid between gas and solids and resistance to heat flow into or through the solids. **K. Panwar et al. [10]** further continued the research in field of CFD analysis by providing the comparative study between results predicted by the CFD simulation of a cylindrical pebble bed thermal regenerator and results predicted by a mathematical model to simulate the operational behaviour of a pebble bed thermal regenerator. In their paper a cylindrical regenerator model of 7ft length and 8 in diameter was used to simulate the behaviour of heat regenerator. Hot flue gases are made to enter at 2000C at an average flow rate of 365 l min⁻¹ made to pass through the regenerator. The mathematical model was developed by energy balance along an elemental volume and the results were compared with the

results obtained by Ansys fluent software. CFD simulations of the model showed the excellent accuracy and negligible deviation with the simulation results of mathematical model. The paper concluded that the simulated CFD model can be used by the designer to reliably design a thermal heat regenerator system for waste heat recovery from flue gases.

In continuation **K. Panwar and D. S. Murthy [11]** in their paper studied the design and thermal characteristics of pebble bed thermal regenerators with small particles with the help of mathematical modeling of regenerator, using MATLAB software to evaluate various parameters of design of the pebble bed regenerator. They calculated Regenerator length, switching time, thermal mean residence time and various other design parameters for maximum single-pass efficiency, maximum heat storage factor and maximum thermal efficiency. A comparison between the co-current and counter current flow within the regenerators was also done in the paper. The paper concluded that the single pass efficiency (i.e., the fraction of heat recovered from the hot-gas stream) of pebble bed regenerator can be improved by increasing length (capital cost) at the expense of a reduced heat-storage factor. Regenerators which are working on symmetric co-current operation their optimal efficiency is obtained at $\tau_s = 1$. It was seen that without any redesign, just by changing the switching time to optimal value, we can improve efficiency of regenerator drastically. The comparison between concurrent and counter current showed that in co-current operation, bigger does not necessarily mean better, the efficiency was doubled by reducing the regenerator length. Without any redesign, just by changing the flow direction, efficiency can be increased. At last most recently **K. Panwar and D. S. Murthy [12]** in their paper investigated the performance of the ball packed-bed regenerator with the help of Ansys fluent software. The study of performance of regenerator included the study resistance and thermal characteristics. In the paper the resistances characteristics of the regenerator based on experimental data were studied and the values of coefficients of Ergun's equation calculated by simulation was compared with the experimental results and a revised Ergun's equation was achieved for ball packed regenerator. In the second part of the paper the thermal characteristics of the regenerator are studied by using 3-D transient model in fluent and the finding were discussed. The heating cycle starts with flue gas entering the regenerator at 1493 K and cooling cycle with air entering the regenerator at 293 K. The variation of pressure drop, temperature, effectiveness, velocity along the regenerator height was studied in the paper.

3 CFD ANALYSIS

Physical model of a Fixed-bed regenerator with cross-section and other geometrical dimensions as given in Table 3.1 was modelled in Ansys Design Modular. Figure 2 shows the physical mode of thermal regenerator.

The complete physical model consists of three main zones namely:

- 1) Flue gas inlet/ Air outlet zone
- 2) Flue gas outlet/ Air inlet zone
- 3) Regenerative zone

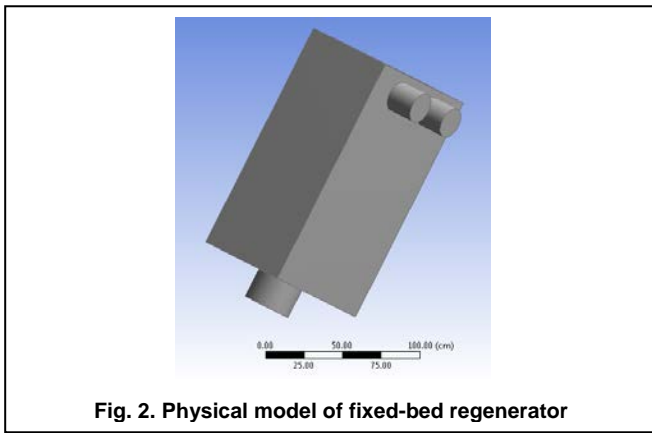


Fig. 2. Physical model of fixed-bed regenerator

At flue gas inlet zone hot flue gases at 500K were made to enter at a velocity of 0.618 m/s (Re=10000) . The flus gases passes through the regenerative chamber were the solid takes the heat from the flue gas and get heated up. From flue gas outlet zone the flue gases at lower temperature are made to exit. The porosity of regenerative chamber was kept 0.4 for optimum working of regenerator. Figure 3.3 shows the geometrical details of physical regenerator model.

3.1 Governing Equations

The governing equations with which the physical phenomenon of flow through porous medium can be governed are as follows :

Continuty Equation :

$$\nabla \cdot (\epsilon \rho \vec{v}) = 0 \tag{1}$$

Momentum Equation :

$$\frac{\partial}{\partial t} (\epsilon \rho \vec{v}) + \nabla \cdot (\epsilon \rho \vec{v} \vec{v}) = -\epsilon \nabla p + \nabla \cdot (\epsilon \tau) + \epsilon B f - \left(\frac{\mu}{\alpha} + \frac{C_2 \rho}{2} \right) \vec{v} \tag{2}$$

$\left\{ \begin{array}{l} \text{Rate change} \\ \text{of momentum} \\ \text{per unit} \\ \text{volume} \end{array} \right\}$	$\left\{ \begin{array}{l} \text{Net rate of} \\ \text{momentum} \\ \text{flux due to} \\ \text{convection} \end{array} \right\}$	$=$	$\left\{ \begin{array}{l} \text{Pressure} \\ \text{force} \end{array} \right\}$	$+$	$\left\{ \begin{array}{l} \text{Viscous} \\ \text{force} \end{array} \right\}$	$+$	$\left\{ \text{Body force} \right\}$	$-$	$\left\{ \begin{array}{l} \text{viscous \& inertial} \\ \text{drag force by} \\ \text{the pore wall} \\ \text{on the fluid} \end{array} \right\}$
--	--	-----	---	-----	--	-----	--------------------------------------	-----	---

The viscous loss coefficient and the inertial loss coefficients of the porous zone are calculated with the help of following equations:

$$\alpha = \frac{(d_p^2 \times \epsilon^3)}{203 \times (1 - \epsilon)^2} \tag{3}$$

where,

d_p is the diameter of solid particle (alumina)

ϵ is the porosity of the regenerator bed

α is the permeability

Viscous loss coefficient in each component direction = $1/\alpha$

$$C_2 = \frac{3.9(1 - \epsilon)}{d_p \times \epsilon^3} \tag{4}$$

where,

d_p is the diameter of solid particle (alumina)

ϵ is the porosity of the regenerator bed

C_2 is the inertial loss coefficient

3 RESULTS & DISCUSSION

The number of hexahedral elements used for meshing the computational model was 281487 with 52003 nodes to capture the effect of stratification more accurately inside the fixed-bed regenerator. To model of fluid flow and heat transfer in the regenerator, CFD software (ANSYS FLUENT), was used. The porous media property is set in the regenerative chamber. The segregated independent and 3D unsteady solver is adopted. Governing equations combined with initial and boundary conditions are discretized in implicit form using the control volume method and solved using SIMPLE-based approach with k-ε turbulence model.

Results have been presented in the chapter for the cases:

- Predicted CFD simulation results compared with experiment results of Ying Liu *et al.* (2014).
- Velocity and temperature variation of flue gases along the regenerator height.
- Pressure drop prediction by simulation for Fixed-bed regenerator.
- Comparison of CFD simulation with Ergun’s Results.

The operation involves the heating of packed-bed at different flue gas velocities as given in paper of Y. Liu *et al.* (2014) i.e. 0.618 m/s and investigating the temperature at different bed heights at inlet and at various sections at 0.25m, 0.5m, 0.75m, 1m, 1.25m and outlet. Fig. 3 shows comparison between the CFD simulated result and previous experimental results for cooling cycle at different heights of fixed-bed regenerator with the bed porosity 0.4. The continuous line in the Fig. 3 shows the simulated CFD results whereas the discontinuous or dashed lines are the plots from the experimental results of Y. Liu *et al.* (2014).

Following assumptions were made to carry out the CFD simulation:

- Intra-particle conduction is neglected.
- The effect of radiation is neglected.
- The regenerator wall is insulated.

Since very good agreement was found between the CFD predicted results and experimental results, it is concluded that the present computational model can be used for further thermal and resistance analysis of fixed-bed regenerator.

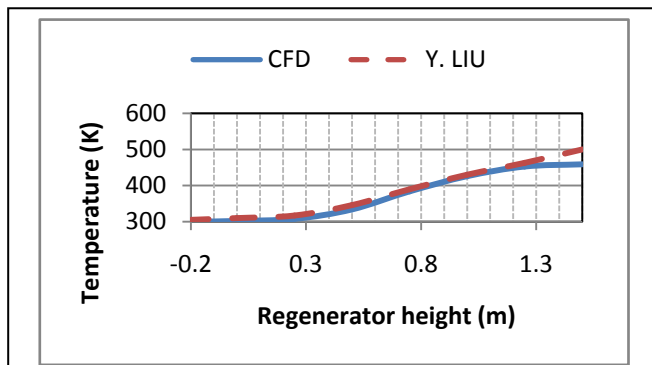


Fig. 3. Validation of CFD results with experimental results of Y. Liu et al. (2014)

Heating cycle was started with entering flue gases at the inlet section (bottom) at a temperature of 500K. Figure 4 show the velocity vector coloured by temperature along the regenerator height. It is clearly observed from the colour of velocity vector that as the flue gas passes through the regenerator it gets colder as it transfers its heat to the regenerative chamber (solids). Figure 4 shows that the regenerative chamber of regenerator is almost 50 % charge after $t = 15$ mins.

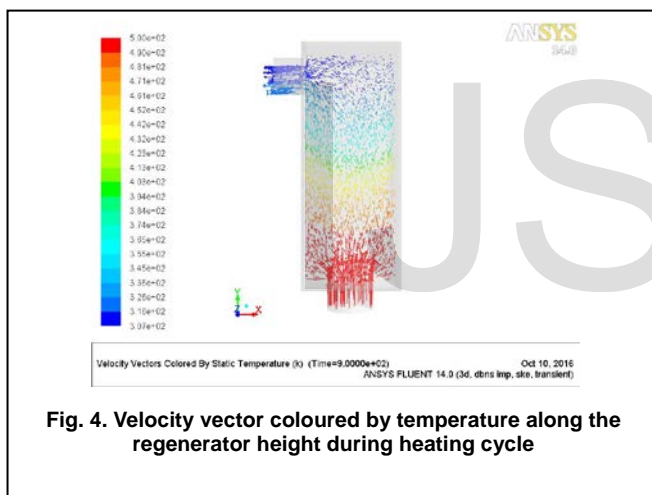


Fig. 4. Velocity vector coloured by temperature along the regenerator height during heating cycle

In various previous 2-D studies published by lot of researchers on regenerators which has been already discussed in the literature review, always assumed that the temperature variation in perpendicular direction to the flow to be constant. But in real problems the temperature variation in the plane perpendicular to the flow of flue gas is not uniform and it is evident from Fig. 5 Hence it concluded that the present 3D CFD model is very much accurate in predicting the thermal and resistance characteristics of fixed-bed regenerators. Figure 4.3 shows the temperature contours at $z = 0, 0.25, 0.5, 0.75, 1.0, 1.25$ m respectively.

Figure 6 shows the temperature variation along the height of regenerator during heating cycle at $t = 15$ mins.

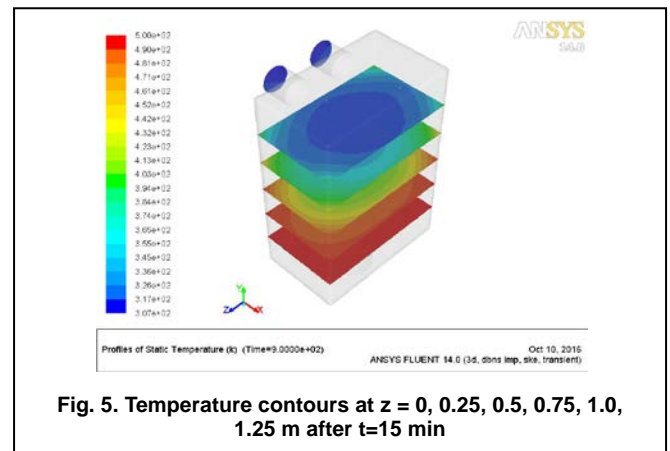


Fig. 5. Temperature contours at $z = 0, 0.25, 0.5, 0.75, 1.0, 1.25$ m after $t = 15$ min

Flue gases at 500K are made to enter the regenerator from the bottom of the regenerator and the heat carried by the flue gases is transferred to the solids in the regenerator bed. After 15 minutes the bed is almost 50% charged. Flue gases are made to exit from the top outlet and at a low temperature since the heat has been transferred to the solids.

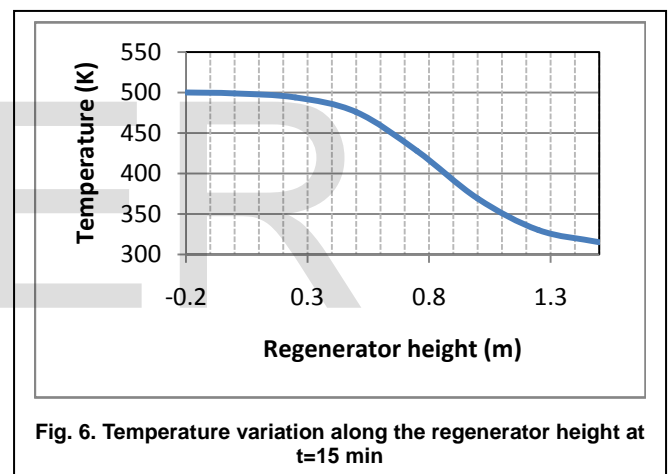


Fig. 6. Temperature variation along the regenerator height at $t = 15$ min

4 CONCLUSION

The thermal characteristics of regenerator were studied and effect of various factors such as regenerator height, D/d_p ratio, and porosity was studied. With the help of Ansys Fluent, contours of pressure and temperature were studied for regenerator with different length. Regenerator was modelled in Fluent and simulations were carried out to study the variation of temperature along the regenerator length. The exit flue gas temperature was calculated from the results of simulations and with the help of it the effectiveness of regenerator was calculated as a function of exit flue gas temperature.

At last the transient CFD models of regenerator with $D/d_p > 15$ was modelled and solved in fluent. The simulation results of each time cycle (heating and cooling cycle) were analysed and efficiency for each regenerator was calculated. This transient model of regenerator can be used for other applications of thermal regenerators

REFERENCES

- [1] K. Panwar and D. S. Murthy, "Analysis of Thermal Characteristics of the Ball Packed Thermal Regenerator". *Procedia Engineering*, vol. 127, pp. 1118-1125, 2015.
- [2] S. Ergun, "Fluid flow through packed columns". *Chemical Engineering Progress*, vol. 48, pp. 89-94, 1952.
- [3] R. E. Hicks, "Pressure Drop in packed bed of spheres". *Ind. Engg. Chem. Fund.*, vol. 09, pp. 500, 1970.
- [4] D. Handely and P. J. Heggs, "Momentum and Heat Transfer Mechanism in regular shaped packings", *AICE. J.*, vol. 9, pp. 590-598, 1968.
- [5] F. MacDonald, M. S. Ei-Sayed, K. Mow, F. A. L. Dullien, "Flow through porous media-Erguns equation revised", *Ind. Engg. Chem. Fund.*, vol. 18, pp. 199-208, 1979.
- [6] B. Eisfeld and K. Schnitzlein, "The influence of confining walls on the pressure drop in packed beds", *Chemical Engineering Science*, vol. 56, pp. 4321-4329, 2001.
- [7] W. Reichelt, "Zur Berechnung des Druckverlustes einphasig durchströmter Kugel- und Zylinderschüttungen", *Chemie-Ingenieur-Technik*, vol. 44, pp. 1068-1071, 1972.
- [8] K. Panwar, D. S. Murthy, H. P. Gangwar, R. Kumar, "Comparative study of mathematical model and CFD simulation of a pebble bed heat regenerator", *International Journal of Scientific and Engineering Research*, vol. 6, No. 4, pp. 1891-1896, 2015.
- [9] K. Panwar and D. S. Murthy, "Analysis of Thermal Characteristics of the Ball Packed Thermal Regenerator". *Procedia Engineering*, vol. 127, pp. 1118-1125, 2015.

IJSER