Performance investigation of a rotary heat exchanger installed with four ducts using computational fluid dynamics

Saneesh Nair[1], *Vinod* Kotebavi[2 *] and *Alokekumar* Das[3]

[1,2] Department of Mechanical Engineering, Amrita School of Engineering, Bengaluru, Amrita Vishwa Vidyapeetham, India
[3]Sr. Manager, Volvo Group India Pvt. Ltd., Bangalore, India

Abstract. The performance characteristics of a commercially available rotary heat exchanger or were determined using CFD. A rotary air pre-heater of a thermal power plant (fuelled by natural gas), situated in Ramin, Iran, was numerically modelled with full scale dimensions using 3D approach. This model was then validated against the experimentally measured values. After a satisfactory validation of the CFD model, a design modification was applied to the model by modelling the heat exchanger with four ducts, with hot and cold fluid ducts arranged alternatively. This was done to improve the heat the heat transfer from hot gases to cold air. The medium of heat exchange (core) is modelled as porous media, since the core in the actual equipment has many number of fine passages arranged in complex manner, which would make the modelling difficult. The model was subjected to same boundary conditions as the validation model. The temperature of the outlets and effectiveness of both the models were compared and was found out that the 4-ducts model is around 12% more effective than the base model.

Keywords: Waste heat, Rotary heat exchanger, computational fluid dynamics, effectiveness

1 Introduction

As the world is seeing rising prices in the fuels, the need to save energy and utilise the lost energy has reached a high importance. In power plants mainly, there is a lot of scope of extracting waste heat from the exhaust gases. There are a lot of systems and equipments which are used for this purpose.

In industries, there are many heat recovery units such as recuperators, regenerators, rotary heat exchanger [1], economizer, heat pumps, etc. In small scale, systems like thermoelectric generators (TEG) [2], PV cells, piezoelectric devices, etc. are used to harvest energy.

The present study focuses on rotary heat exchanger. A rotary heat exchanger is a heat exchanging device used within the supply and exhaust streams of a gas handling system. In power plants, it is used for the heat recovery purpose from the exhaust gas coming out of the boiler.

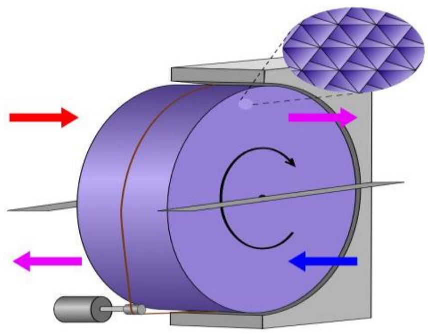


Figure 1. Rotary Heat Exchanger

As shown in the Figure 1, it consists of a rotating thermal wheel (also called core) which transfers the heat taken from hot fluid to cold fluid. The core of the heat exchanger consists of a honeycomb matrix and is made up of heat-absorbing material, which is slowly rotated. As the core rotates, exhaust gas stream transfers its heat to the fresh air stream through the core. The performance of the core depends on the thermal gradient between the two streams.

Rotary regenerators are being used very commonly used in recent decades due to its compact design, lower

* Vinod Kotebavi: k_vinod@blr.amrita.edu

operating cost in terms of heat transfer per unit volume and easy control of the heat exchange by controlling the heat wheel speed [3]. It is the one of most effective heat exchanger to extract waste heat in power plants and to cool down the electronic equipments in industries and data centres.

In one research, authors have conducted a study on the use of the rotary heat exchanger in a data centre and concluded that these regenerators are highly dependent on climate [4]. In another research, authors have used it in C.I engine in order to extract waste heat from 653K exhaust gas and concluded that a significant amount of waste heat can be extracted from the engine exhaust [5].

Studies have also indicated the rotary heat exchanger is more effective than plate type heat exchanger [6]. Some researchers have also attempted to subject the heat wheel to a varying temperature data and to study the performance of heat wheel under such conditions [7]. The inlet condition was mathematically modelled. It was found that outlet conditions also followed the same trend as the inlet conditions.

According to the studies conducted, the performance of the thermal wheel is limited to only half of its travel through the cold fluid duct, i.e. if we study the outlet temperature profile of the cold fluid, we would observe that there is a significant change in outlet temperature of the cold fluid only in the first half portion of the cold fluid duct. The second half of the cold fluid duct has the temperature close to its inlet [8].

In an attempt to solve this limitation, the current study is focused to determine how the preheater would perform if it is installed with four ducts, instead of two, with hot and cold fluid duct arranged alternatively. Clearer understanding can be achieved from the figure 8. At the end of the simulation, the performance of both the base model and '4-Ducts' model were compared on the basis of respective effectiveness.

The mathematical model of the effectiveness was done with following assumptions:

- Mass flow rate of both the streams are same, ($M_h = M_c$).
- Knowing the properties of exhaust gas respective to Ramin power plant, Iran, is a difficult task. Thus to simplify the problem, air was chosen as the standard gas for both the streams and was different physical conditions.
- Since, there is negligible difference between the specific heat of the host exhaust gases and the cold air, thus they were assumed as equal, ($C_{pa} = C_{pg}$).

$$\epsilon = \frac{Q_{act}}{Q_{max}} = \frac{M_h\, C_{pg}\,(T_1 - T_2)}{(MC_p)_{min}(T_1 - t_1)} = \frac{M_c\, C_{pa}\,(t_2 - t_1)}{(MC_p)_{min}(T_1 - t_1)} \quad (1)$$

Where,

Q_{act} = actual heat transfer

Q_{max} = maximum heat transfer

M_h , M_c = hot & cold mass flow, respectively

C_{pa} , C_{pg} = specific heat of cold air and hot exhaust gas, respectively

$(MC_p)_{min}$ = minimum heat capacity

T_1, T_2 = inlet & outlet temperature of hot exhaust gas, respectively

t_1, t_2 = inlet & outlet temperature of cold air, respectively

Thus, considering the above assumption, above equation can be modified as

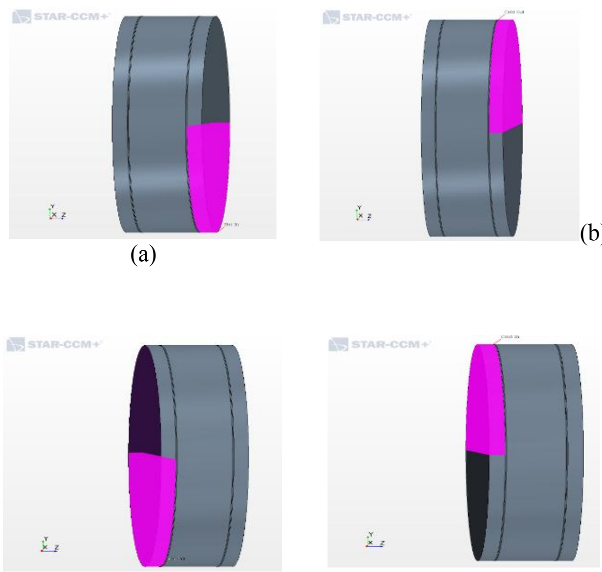
$$\epsilon = \frac{(T_1 - T_2)}{(T_1 - t_1)} = \frac{(t_2 - t_1)}{(T_1 - t_1)} \quad (2)$$

2 Modelling & meshing

The CAED modelling and meshing of the CFD model is done in Star CCM+, using dimensions mentioned in [1]. The core diameter is 9864 mm and the ducts are arranged as shown in Figure 2(a) – 2(e) below.

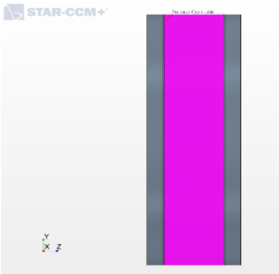
2.1 Base Model

The Figure 2(a)-(e) shows the CAED model of the current rotary heat exchanger. The model consists of five important parts namely; cold air inlet & outlet, hot exhaust gases inlet & outlet and the core. Core is the part where the heat transfer phenomenon takes place as the part rotates and stores the heat from the hot exhaust gases, locally and then passes it on to the cold air stream.



(a) (b)

(c) (d)



(e)

Figure 2. (a)Hot Fluid Inlet (b)Cold Fluid Outlet (c)Hot Fluid Outlet (d)Cold Fluid Inlet (e) Core

The meshing of the model is done using Surface Remesher and Polygonal Mesher. The outlets of the ducts have been extruded to some distance using Extruder Mesher. This is done to create a stabilised flow through the ducts. The Table 1 represents the mesh parameters and sizes that were set to generate a suitably fine mesh for efficient computation. Figure 3 shows the meshed base model.

Table 1. Mesh Parameters

Parameters	Values
Base Size (mm)	100
Minimum Surface Size (mm)	40
Target Surface Size (mm)	40
Extrusion length (mm)	±2500
No. of layers	10

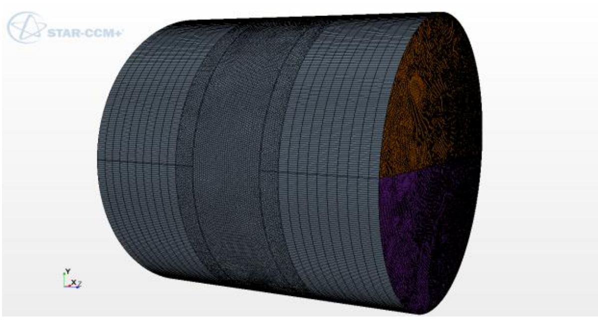


Figure 3. Meshed Base Model

2.2 4-Ducts model

In an attempt to use most of the heat energy from the waste flue gas, the rotary heat exchanger was modified as shown in the Figure 4 below. Figure 4 shows the hot end of the 4-ducts model. The orange sections represent cold fluid outlet and the purple sections represent the hot fluid inlet. The same meshing phenomenon and the meshing parameter (as mentioned in Table 1) were used for the 4-Ducts model. The meshed model is as shown in the Figure 5.

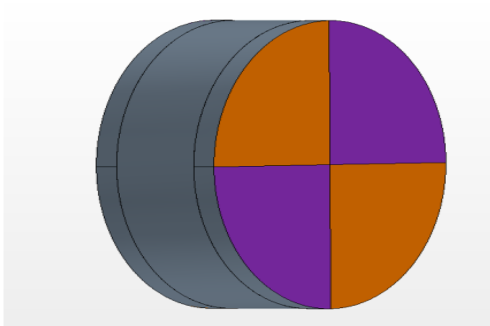


Figure 4. 4-Ducts Rotary Heat Exchanger CFD Model

Figure 5. 4-Ducts meshed model

3 CFD Analysis

The validation and the 4-Ducts model were subjected to implicit unsteady analysis, with inlet and outlet boundary conditions set as stagnation inlet and pressure outlet respectively. The boundary parameters were set as shown in the Table 2.

Table 2. Boundary parameters

Revolution Speed	2 rpm
Inlet Air Temperature	348.9 K
Inlet Air Pressure	2970 Pa
Outlet Air Pressure	1650 Pa
Inlet Gas Temperature	634.2 K
Inlet Gas Pressure	-1890 Pa
Outlet Gas Pressure	-3110 Pa

Since, the modelling of honeycomb structures or the fine passages through the heat transfer medium is difficult, the core was treated as a porous region with a porosity of 0.8[7].

Also, the rotation of the model was induced using Rigid Body Motion (RBM) technique, in which whole of the mesh is moved by the solver in order to determine the rotational effects. Rotation rate was set at 2 rpm [1].

3.1 Validation Model

The validation of the base model was done against the experimental results got in [1] using the above mentioned models and boundary conditions. The results were in close validation to the reference experimental values as shown in the Table 3.

Table 3. Reference validation results

Results	Outlet Gas Temperature	Outlet Air Temperature
Experimental Results	393.4	534.3
Numerical Results	430.5	520.6
Error (%)	8.6%	2.6%

Figure 6 shows the temperature profile at the hot end, i.e. at the end where cold fluid exits and hot fluid enters. The absolute red region of the temperature profile represents the entry of the hot fluid. The varying temperature profile represents the cold fluid exit. As can be seen clearly from the Figure 6, only first half of the cold fluid is heated and other half is at almost the same temperature as at entry (refer the colour map).

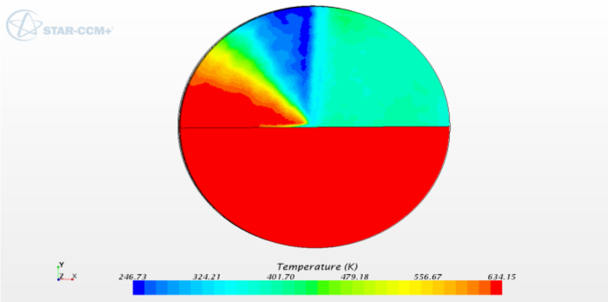


Figure 6. Hot end Temperature Profile

The Figure 6 also indicates some regions show unreasonable temperature profile (blue regions). This is believed to be an error which has occurred due to the reverse flow effects in some cells. This issue can be rectified using finer mesh sizes, but it would require more amount of computational time. Thus, temperature value has been taken from the yellow region.

3.2 4-Ducts Model

Simulation of the 4-Ducts model was done with the same physics models and boundary conditions as mentioned in Table 2.

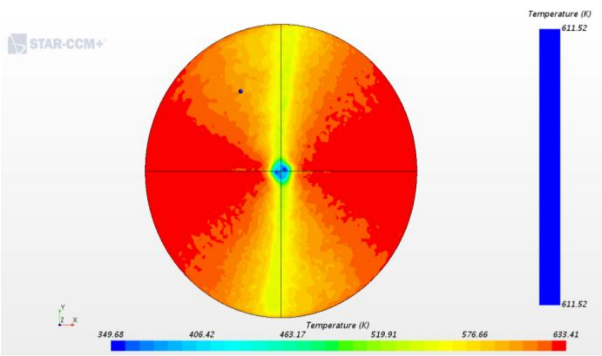


Figure 7. Hot end temperature profile

The Figure 7 shows the temperature profile of a plane placed near to the hot end but inside the heat exchanging medium. It should be noted that in RBM approach, whole of the mesh is moved with the rotational speed specified. When the simulation was stopped, the moving mesh of the heat exchange medium was stopped at position as shown in the above figure. To correct this condition, would again be requiring a lot of computational time.

However, it was deduced form the Figure 7 that the absolute red regions belong to the hot fluid inlet and the regions with colour variation (shades of blue and orange) belong to the cold fluid outlet. In order to observe the result more clearly, few different methods were adopted which are discussed below.

A point probe was kept inside the cold fluid outlet (blue dot in top left quarter of the Figure 7), which returned the temperature value of the heated air (refer the right vertical temperature bar). Thus, temperature of the cold fluid out was recorded as 611 K.

Another method involved placing a plane through the ducts as shown in the Figure 8 below.

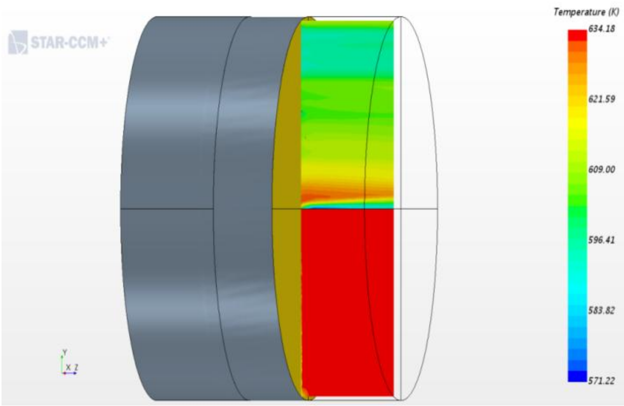


Figure 8. 4-Ducts temperature profile viewed from side

Figure 8 shows the temperature profile on a plane which is placed in the hot fluid inlet and cold fluid outlet. Absolute red region belongs to the hot fluid inlet and region with colour variation (shades of green and yellow) belongs to cold fluid outlet. The Figure 8 indicates that the cold fluid outlet temperature is around 610 K, which is same as observed in the Figure 7.

4 Results & discussions

The effectiveness of both the model was compared using the mathematical model (2). The effectiveness values were found as tabulated in the Table 4.

Table 4. Comparison of the performance of the base model and 4-Ducts model

Results	Outlet Gas Temperature (K)	Outlet Air Temperature (K)	Effectiveness (%)
Base Model Results	430.5	520.6	71.39
4-Ducts Results	395	611.52	83.81

- From Table 4, we can see that there is an appreciable increase in the effectiveness of the rotary heat exchanger by 12.42%.
- If we compare the Figure 6 and Figure 7, we can clearly see that the latter temperature profile shows very less colour variation. This indicates that the outlet temperature of the cold fluid has almost reached the inlet temperature of the hot fluid.
- From Figure 7, the point probe in the top left part of the figure indicates the temperature at that point. The point was placed inside the cold out duct. This indicates that the temperature of cold fluid has reached to around 611 K.
- In Figure 6, indicates that base model has not recovered the heat from the hot fluid as effectively as the 4-Ducts model, since the temperature of cold fluid at the outlet is nearer to its inlet condition.

The Figure 7 shows that the temperature distribution is uniform. Thus, there would be lesser thermal stress on the core material as compared to the base model.

5 Conclusions

The results obtained reflect a significant improvement over the base model. The outlet temperature of the cold fluid was greatly increased to 600 K. The outlet temperature of the hot fluid was also significantly reduces to around 395 K, which showed that more of the waste heat has been recovered as compared to the base model. Though there were reverse flows during the simulation, the end result was satisfactory.

- Such a great increase in the outlet temperature of the cold fluid establishes that it would require less fuel in the boiler. Thus the boiler efficiency would be greatly increased, reducing the load on the boiler.
- There would also be a significant reduction in the fuel consumption rate, for the same amount of the turbine output.
- If the fuel consumption is kept constant, then the gas can be superheated to a greater extent. This would increase the turbine power output significantly.
- Also, it can be deduced that there is a great increase in energy recovery which usually goes wasted with the exhaust gas.
- Above conclusions clearly represents that, if 4-Ducts rotary heat exchangers are used in a thermal power plant, there would be a great increase in the power plant efficiency.

References

1 Heidari-Kaydan, Armin, and Ebrahim Hajidavalloo, *Applied Thermal Engineering* 73.1 (2014): 399-407.

2 Rohit, G., D. Manaswini, and Vinod Kotebavi, *AIP Conference Proceedings*. Vol. 1859. No. 1. AIP Publishing, 2017.

3 Worsøe-Schmidt, P, *International Journal of Refrigeration* 14.4 (1991): 233-239.

4 Mok, Seungho, et al., *2016 15th IEEE Intersociety Conference on*. IEEE, 2016.

5 Bhojane, Monali S., and M. S. Deshmukh, *Emerging Trends in Engineering and Technology (ICETET), 2010 3rd International Conference on*. IEEE, 2010.

6 Swapnil Nilange, Bhavesh Kakulte, Bhushan Maheshwari and Jagdish Patil , *International Journal of Research in Engineering and Technology (IJRET) April 2013*, Volume: **02** Issue: 04 : 2319-1163

7 Fathieh, F., et al, *International Journal of Heat and Mass Transfer* 87 (2015): 312-326.

8 Vishal Hegana and P.R. Kulkarni, *International Journal of Advance Research In Science And Engineering (IJARSE) May 2015*, Vol. No.4: 2319-8354(E)