International Journal of Power and Energy Systems, Vol. 43, No. 10, 2023
An Open Access Paper

CFD ANALYSIS OF 210 MW BOILER'S ECONOMISER FOR DETECTING AREA AND CAUSES OF TUBE FAILURE

Chandrakishor L. Ladekar,* Manoj R. Dahake,** Chandrakant B. Kothare,*** and Ajay V. Kolhe****

Abstract

The forced outages in the thermal power plants are due to boiler tube leakage (BTL) as they run at full load, which affects the power plant's performance. The areawise and causewise boiler tube failure data analysis showed more percentage failure in the finned tube economiser due to erosion and overheating of the tubes. Computational fluid dynamics (CFD) modelling can be applied in the power plant economiser. This work focuses on CFD analysis of a finned tube economiser to see velocity, temperature, and pressure at various places in the economiser. A comparison of simulated velocity with cold air velocity test data showed that actual velocity along the water wall is approximately 10–20% more than measured by simulation. Further investigation in close view shows that the tube failure occurs at U-bends due to ash particles' high velocity and temperature. The low temperatures at the corners of the attached fin resulted in the welded corner's failure, further validated through the tube failure data. A low temperature on the downside leads to pitting damages. Pressure variations show the low pressure on the downside of the tube, resulting in the un-uniform pressure of the fluid particles in close vicinity to the tubes of the economiser and a reduction in the heat transferring area. Redesigning the fin structure over the tube is suggested to eliminate this problem. High static temperature and velocity were observed near the walls. This helps to identify the location and magnitude of the temperature and velocity for further modification. The modification in terms of re-designing the fins over the tube, placement of the baffles in the flow passage, and introduction of the mixing chamber and blowers were suggested based on the study. The impact of the flue gases can be predicted using the advanced technique of life-placing baffles, which will expect the tube life based on the simulation model and can reduce the cost of experimentation.

* Pimpri Chinchwad CoE (PCCoE) Pune & S. P., Pune University, Pune, India; e-mail: chandrakishor.ladekar@pccoepune.org
** AISSMS CoE, Pune & S. P., Pune University, Pune, India; e-mail: mnjdahake@gmail.com
*** Shree Shankarprasad Agnihotri College of Engineering, Wardha, India; e-mail: chandrakant.kothare@rediffmail.com
**** Kavikulguru Institute of Technology and Science (KITS), Ramtek, India; e-mail: ajay.kolhe4@gmail.com
Corresponding author: Chandrakishor L. Ladekar

Recommended by Jen-Hao Teng
(DOI: 10.2316/J.2023.203-0484)

Key Words

Flow analysis, CFD analysis, boiler economiser, erosion failure, boiler tube leakage

1. Introduction

In many developing countries like India, coal is the primary fuel for power production. As per the report by India's coal ministry, it accounts for 55% of the country's energy needs [1]. As per the Central Electricity Authority (CEA) report in June 2022, coal-based power plants produced 204.8 Giga Watt (GW) of power, contributing 50.7% out of 437.6 GW total generations [2].

The assessment of the performance of India's thermal power plants for 2016–17 was prompted by data showing that the percentage of a generation lost due to forced outages (FO) rose to 24.52% in that year from 21.91% the year before. A rise in reserve shutdown (RSD) and coal/transmission restrictions have led to more FO. In addition, India's plant load factor (PLF) for its thermal power units fell to 59.06% in 2016–17, down from the 61.06% recorded the year before. RSD caused an increase in generation loss, contributing significantly to the decrease in PLF. Tube leakages constitute a significant cause of FO in power plants in developing nations. Tube leaking is a complex problem with a variety of reasons and solutions. Fuel efficiency and power output may be improved by looking into the causes of tube leakage and fixing them [2]. Table 1 shows the year-by-year area/cause of FO and energy loss (EL) occurrence for 143 units of 210 MW.

This work was carried out to solve the force outage problem of Maharashtra State Power Generation Company (MSPGCL) as part of the boiler tube leakage (BTL) project. Particularly, the data of Bhusawal Thermal Power Station (BTPS) 2 units of 210 MW was used for the study. To find the significant area and cause of BTL, data of the causewise and areawise BTL failure instances of MSPGCL in all 210 MW units are used to focus on the particular area. They are given in Tables 2 and 3, respectively.

Causewise and areawise failure trends in 2 units of a 210 MW Power Plant at BTPS were studied and analysed for 3 years performance. Causewise, failure data shows

Table 1
The Area/Cause of FO and EL for 210 MW Units in 2014–15 to 2016–17 [2]

Area/Cause of the Outage	FO in Number			Millionth Units EL			F.O. Loss in %		
(210 MW Cap. Group)	14–15	15–16	16–17	14–15	15–16	16–17	14–15	15–16	16–17
1. Water wall	309	237	150	4059.1	4457.5	3710.8	10.28	7.67	5.04
2. Superheater	36	37	50	686.62	655.06	511.93	1.74	1.13	0.70
3. Reheater	35	46	64	589.14	549.7	1547.3	1.49	0.95	2.10
4. Economiser	66	97	102	941.16	1014.8	1200.3	2.38	1.75	1.63
5. Air preheaters	16	23	20	220	165.7	255.45	0.56	0.29	0.35

Table 2
Causewise Number of BTL Failure Instances in 2 Units of 210 MW Power Plant at BTPS [2]

Causes	14–15	15–16	16–17	Total
Erosion	4	2	1	7
Stress corrosion	0	0	0	0
Water corrosion	0	0	0	0
Dissimilar joint failure	0	0	0	0
Overheating	2	2	2	6
High-pressure joint failure (HPJF)	0	1	0	1
Freeting	0	0	0	0
Air ingress	1	0	0	1
Total	7	5	3	15

Table 3
Areawise Number of BTL Failure Instances in 2 Units of 210 MW Power Plant at BTPS [2]

Zone	14–15	15–16	16–17	Total
Water wall	1	1	1	3
Economiser	1	3	2	6
Reheater (RH)	3	1	0	4
Superheater (SH)	1	0	0	1
Low-temperature superheater (LTSH)	1	0	0	1
Total amount	7	5	3	15

significant erosion problems followed by overheating and HPJF. Areawise failure trends for the number of failures in components, such as waterwall, superheated, reheater, and economiser, show that the economiser tube failure is the highest in which maximum loss occurs. Therefore, the study focuses on analysing the reasons for the economiser's failure and finding remedies with a specific focus on erosion and overheating failure to solve the problem of BTL.

Literature related to leaking detection and fault finding suggested various methods, such as acoustic signal processing to detect leaks in pressurized systems of industrial power plants and microstructural analysis of failed tubes [3], [4]. However, such techniques are expensive and time consuming as well are adopted either after failure or during boiler operations. Identifying the problems and visualising the flow in operational boilers is difficult. Due to its ability to speed up the design process compared to experimental testing for data collecting, computational fluid dynamics (CFD) software has seen a significant uptick in its use. Although a few variables can be monitored simultaneously in real-world testing, a CFD study can measure them in high resolution over space and time [5]. It is modern and economical to visualise the flow pattern, determine various process parameters, and detect areas and causes of tube failure in the boiler, as seen in multiple kinds of literature. 2-D analysis of superheater tubes using CFD tools helps in identifying high temperature affecting zones, which enables the analysis of highly complicated structures while considering specific three-dimensional geometrical models [6]. Fly ash utilised in boilers is analysed for its high ash content, unburned carbon percentage, and gas temperature [7]. FLUENT was used to predict relative rates of pipe surface erosion [8]. CFX-4 code's applications can be integrated into automated information and process control systems; making them more accessible and valuable [9]. CFD modelling on the air heater's blank tube helped to reduce erosion and corrosion, which may further minimise many boilers' operating and maintenance costs. Enhancing the FURNACE code will reduce gas-phase emissions and improve the bagasse-fired boiler performance [10]. A coal-based boiler's T-22 superheater tube failure study uses numerous tube failure analysis methods [11].

More specific literature on economiser and its fin structure is studied. The attempt of modelling plain, multi-lead rifle (MLR), and longitudinal fin boiler tubes and its numerical analysis carried out using Altair AcuSolve showed a rise in temperatures for longitudinal fins and MLR tubes [12]. The effect of MLR geometries, such as their number, height, and pitch length, on heat transfer in the boiler, was also studied using CFD [13]. Considering the heat transfer enhancement of MLR tubes, the 3D economiser single-column MLR tube's performance was reviewed and attempted to analyse flue gas process parameters using the CFD approach [14], [15]. Using a CFD tool, the performance of helical serrated finned tube bundles in an inline bare tube counter cross-flow water tube boiler economiser was also examined for improved heat transfer [16]. 3D modelling is performed in CREO and ANSYS to determine temperature distribution and heat transfer rate in the econometrics to evaluate the best materials by changing the mass flow rate [17]. The flue gas temperature, pressure, and velocity field of fluid flow using the actual boundary conditions have been analysed using CFD to create a three-dimensional model of the economiser coil. It allows for quickly analysing various design options without modifying the object and provides significantly more data to interpret the results [18].

Many methods for tube leakage detection and identification of the reason for failure are available, like visual observation, location identification, base-alloy composition determination, microstructural analysis, hardness evaluation, fractographic analysis, and X-ray diffraction analysis. CFD has excellent potential to measure desired process parameters at once without experimental data acquisition and analyse a number of modifications virtually in less time and cost involved in experimentation. Therefore, the present work focuses on creating a CFD model of a boiler economiser based on the actual dimensions and carrying out a simulation using accurate plant data. The results of CFD simulation were validated using an experimental tested cold air velocity test (CAVT) and other test data and further enabled to establish a process and design modification in existing boiler economize.

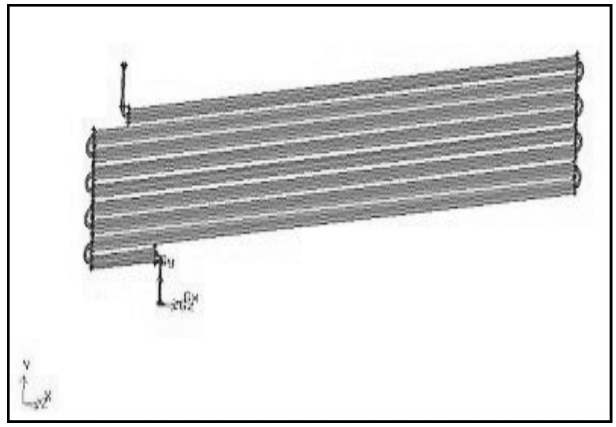


Figure 1. Modelling of finned tube economiser.

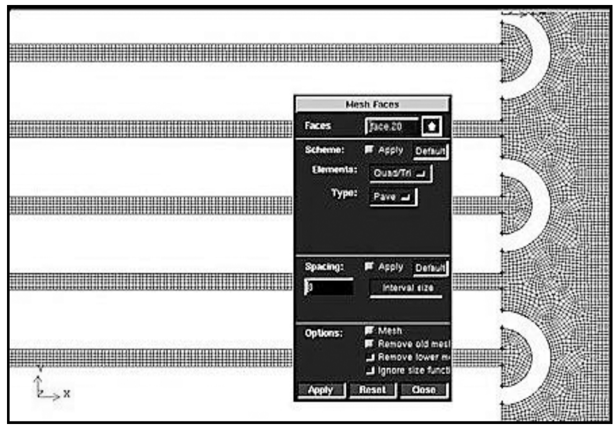


Figure 2. Meshing of finned tube economiser.

2. CFD Analysis of Finned Tube Economiser

The complete CFD analysis is divided into six steps which are as follows: (a) creating economiser geometry in GAMBIT, (b) meshing economiser geometry in GAMBIT, (c) specifying boundary types in GAMBIT, (d) pre-processing (defining boundary condition), (e) solver, and (f) post-processor.

2.1 Creating Economiser Geometry and Meshing using GAMBIT

Finned tube staggered arrangement economiser with a total surface area of 5690 m^2, with a single-stage assembly with 270 tubes. The tube material is SA 210 Gr. A1 with tube diameter and thickness 44.5 mm OD × 4.5 mm thick, respectively. The lateral and parallel pitch is 100 mm and 114 mm, respectively. The fin height is 50 mm, and the thickness is 5 mm. The tube design pressure is 162 kg/m^2 with a filling capacity of 25.13 m^3. The finned tube economiser was modelled using GAMBIT as per the specification and actual drawing of the power plant. The geometry created in Fig. 1 shows the single coil of the finned tube economiser in the 2-D mode. Fins are over the tube up to a specified length. Tubes with a U-tube bend, the inlet, and the outlet of each coil are also shown in the model.

Figure 2 shows the finned tube economiser coil mesh (economiser GAMBIT mesh with 2-D). Here, we adopt the quad/tri cells mesh for complex geometry because of its accuracy over irregular surfaces. The interval size given was 8, which meshes the geometrical model into 489,536 cells.

2.2 Specify Boundary Types and Defining Boundary Conditions (Pre-processing) in GAMBIT

The top edge is the flue gas entrance to the economiser, the bottom edge is the flue gas outlet, and the side edges are the walls. Control panel > zones > specify boundary types. Select VELOCITY INLET for Type, and Select Apply. The new entry should show towards the top of the window beneath the Name/Type field. Repeat this procedure for the remaining three edges, as shown in Table 4.

Once the mesh has been defined, meshed geometry is imported from the mesh file. Then the flue gas properties and peripheral conditions must be applied as the boundary conditions. The properties include flue gas viscosity, density, thermal conductivity, etc. The boundary condition taken to be into consideration is shown in Table 5.

Table 4
Type of Boundary Conditions Applied to Different Edges

Edge Position	Name	Type
Top	Inlet	VELOCITY_INLET
Bottom	Outlet	PRESSURE_OUTLET
Left	Wall	WALL
Right	Wall	WALL

Table 5
Flue Gas Conditions

Parameters	Specification
Inlet temperature	732 K
Outlet temperature	641 K
Inlet velocity from cold air velocity test (CAVT)	1.41 m/sec
Inlet pressure (gauge)	−362.97 Pascal
Outlet pressure (gauge)	−529.74 Pascal

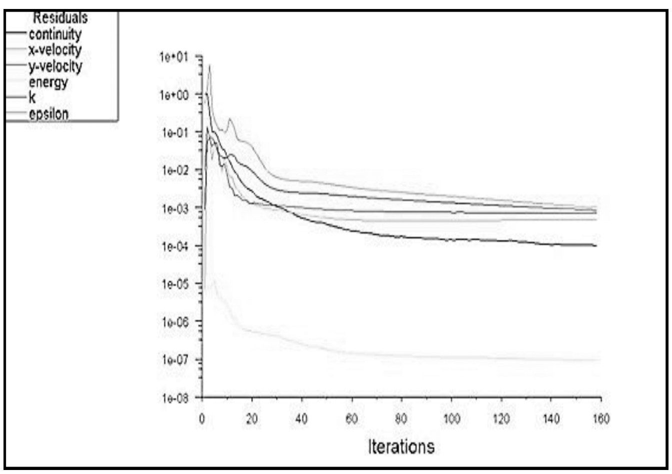


Figure 3. Convergence graph.

2.3 Solver

FLUENT generates a residual for every governing equation that is solved. The residual indicates the degree to which the current solution satisfies the discrete version of each governing equation. Iterate the solution until the residual for each equation is less than $1e^{-6}$. Figure 3 illustrates the convergence graph changes to $1e^{-6}$, the residual under the convergence condition for continuity, x-velocity, and y-velocity. Also, select Plot from the Options menu and repeat until convergence. Perform 100 iterations to initiate the calculation. According to the analysis, residuals for each iteration are produced and plotted in the graphics window.

2.4 Post-processor

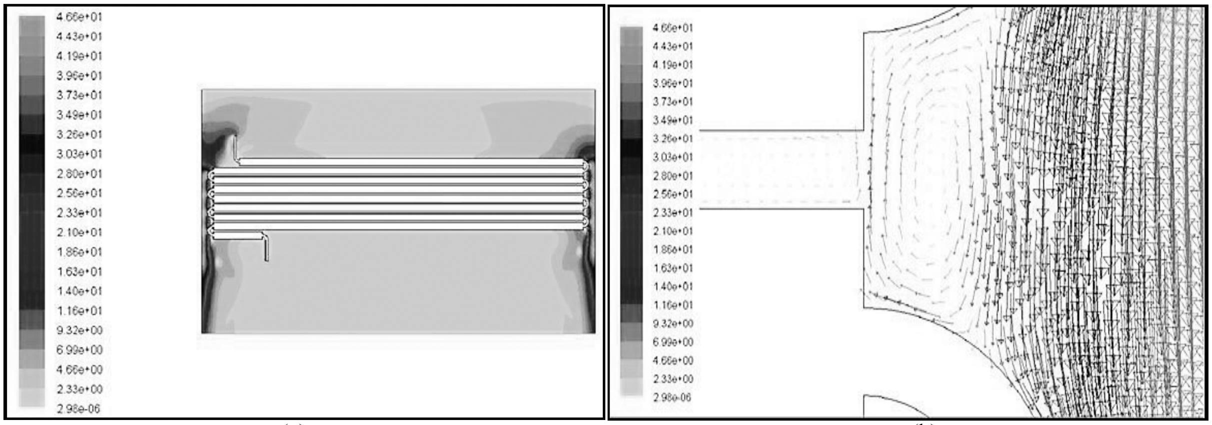
Finally, it can read the data of the post-processor once the flow has successfully converged, which can extrapolate the velocities and temperatures at the grid point to every part of the domain. Graphs typically generated by the post-processor show the variation of a wide range of parameters with time and space. We show the result of CFD solver mode in FLUENT post-processor. The obtained results are affected by the boundary condition specified in the FLUENT pre-processor. Our primary goal is to determine the temperature distribution along the tube coil in the critical zone and the region where velocity exceeds the limiting value. This will be extremely useful in determining the critical area of tube failure and its cause.

2.4.1 Velocity Plots

Economiser's flue gas velocity distribution is given in Fig. 4(a) by considering the zero roughness of the wall. The average velocity is close to 3 m/s, whereas the measured velocity is 3.5 m/s. The velocity near the water wall is 10–20% more than calculated by simulation. Velocity near the wall close to the U-section of the tube bends is nearly 4.5 m/s which is much higher than the average velocity. Significant velocity variation in this region is due to flow directly from the top side towards the wall side and less space between the wall and the tube's U-section. Figure 4(b) shows a close view of velocity vectors and their flow directions. The velocity vector directly comes over the U-tube bend and flows inside along the tube length with decreasing velocity. The velocity near the U-bend top side is 4–5 m/s, whereas the low side has low flue gas velocity in the 2–3 m/s. This led to lower heat convective heat transfer and low heat rate. Further, we observe swirls close to the tube circumference and considerable significant velocity variation over the economiser. Here, we can predict the proper mixing of the flue gases using the modified run. Considerable velocity variation is found in this region near the wall, almost double that of average velocity. The improper mixing at the entry side and higher density towards the wall side as flue gas rush towards the wall and pass through the U-section's end.

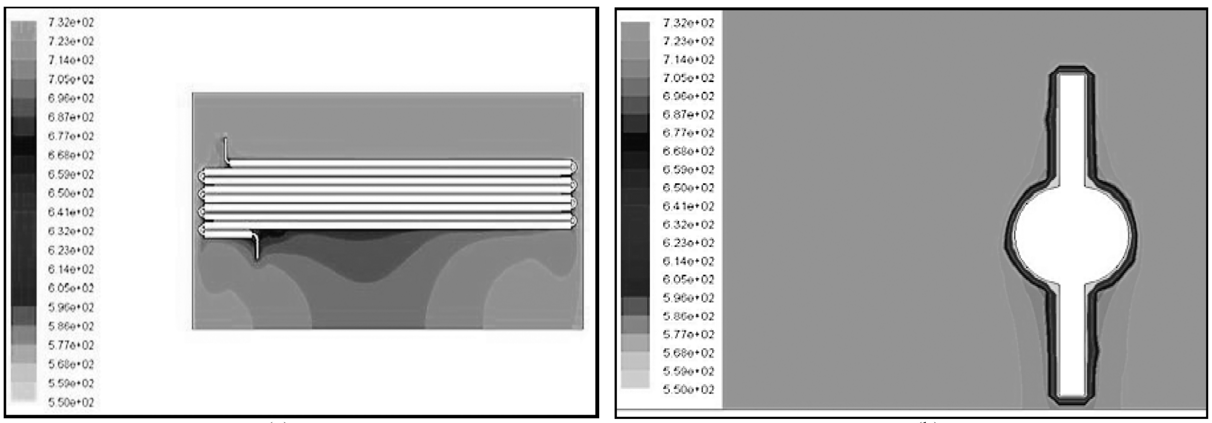
2.4.2 Temperature Plots

The temperature distribution in the finned tube economiser section is given in Fig. 5(a). As the process described, the economiser is the device for heat recovery and cross-flow direction. The temperature at the entry is close to 723 K and uniformly distributed, but towards the end and at the middle section, it is close to 670 K. It is not uniform, and a considerable variation of temperature change from LHS to RHS is observed. The low temperature is on the downside of the economiser. We found that the tube bends are exposed to the high-temperature area. Low temperatures near the economiser's inlet header can cause pitting damage, and sudden temperature changes can cause thermal embrittlement.



(a) (b)

Figure 4. (a) Velocity of flue gases across finned tube economizer; (b) Close view of the velocity vector near the U-tube bend.



(a) (b)

Figure 5. (a) Static temperature of flue gases along finned tube economizer; (b) Static pressure of flue gases along finned tube economiser.

The finned tube's close view is shown in Fig. 5(b). Temperature over the immediate vicinity of the tube is the same in both places, but the different temperatures of the flue gases are different. The temperature at the corners of the welded fins is close to 550 K, a significant temperature difference resulting in higher thermal stresses in this region. Visual inspection of failed tubes in this section found a change of shape and rapture in this area due to high-temperature crip and thermal stress.

2.4.3 Pressure Plots

Static pressure variation over the economiser tube is shown in Fig. 6(a); the pressure from the top to the bottom of the stage increases. We can find certain kinds of pressure variation close to the tip of U-tube bends but within a considerable limit.

Figure 6(b) shows the tube's close view showing the static pressure variation over the tube. We have observed relatively high pressure over the top side of the fin tip and low pressure at the bottom side fin tip. The study of the pressure variation can help remedial action over the process parameter and modification to reduce the tube leakage problem.

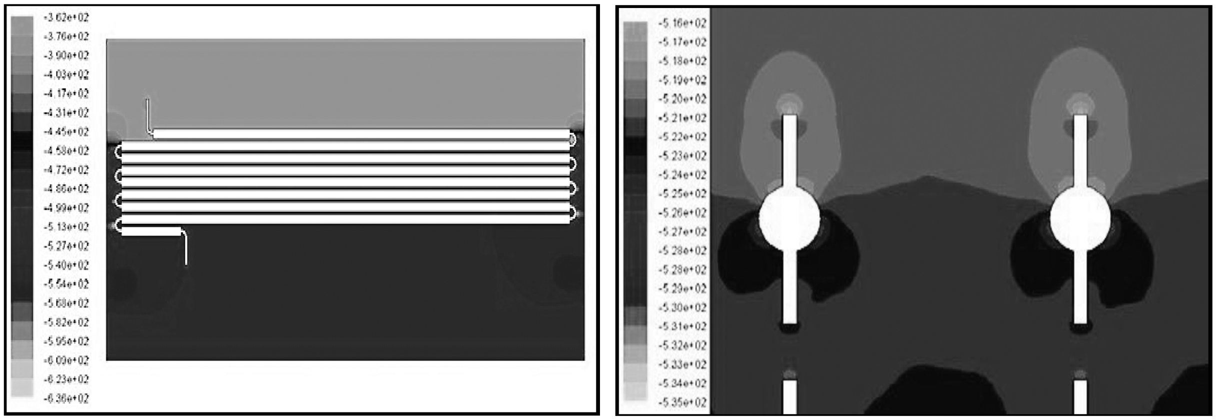
The velocity of the flue gases near the U-tube bend is more than permissible. Also, the temperature plots show that the high temperature affecting zones are topside of the economiser.

3. Results and Discussions

The CFD model results compare with Plant control room data and CAVT data. Figure 7 shows the grid with the iso-lines at the $y = 3000$ mm distance. The coordinates $x = 0$ and $y = 0$ are set at the economiser inlet tube.

Figure 8(a) shows the variation of static temperature versus the position along the x-axis. We observed the V-shape curve, with a sudden drop in the temperature at a distance of approximately negative 150 mm and increases at a negative distance of roughly 50 mm. Figure 8(b) shows the variation of velocity magnitude with the position along the x-axis. The velocity plots show that the rise in the velocity is 6 m/s on the left side, while on the right side, it rises to 4 m/s.

The plant conducts CAVT of the 210 MW boilers frequently. During these tests, they used anemometers to measure local velocities at different planes of the



(a) (b)

Figure 6. (a) Static pressure of flue gases along finned tube economiser; (b) Static pressure of flue gases along a tube of the economiser.

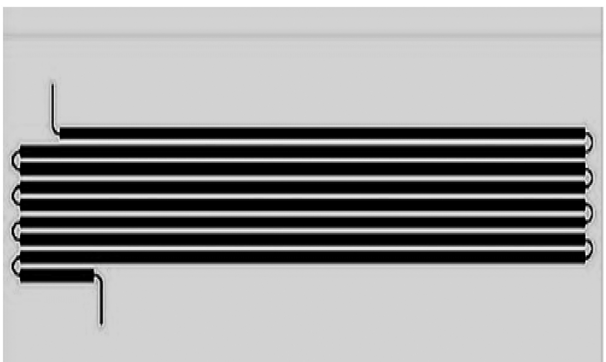


Figure 7. Grid showing the iso-lines at $y = 3000$.

boiler economiser (ALNOR 6000AP and Dwyer Series 471-thermal anemometer). There were seven lines on a plane from the left side to the right-side wall of the boiler on which velocity measurements were carried out. On each line, measures were taken at several points located from the bottom to the top of the front side to the rear. Similar measurements were made in the CFD simulation. A comparison of the predicted velocity profiles with the CAVT data is shown in Fig. 9.

The CFD simulations have captured the key features of the flow reasonably well, except at the rear end. CAVT data showed lower velocities near the rear end than the bulk region, whereas CFD simulations showed the opposite trend. If there is a gap near the rear end, the velocities near the end are expected to be higher than the bulk, consistent with the CFD simulations. The microstructure of the tube indicates that the tube has failed due to short-term overheating. Short-term overheating generally occurs due to the following reasons: (a) flow starvation due to low drum-level operation and (b) the flow restriction is due to blockage or deposition inside the tube. It is recommended that the operational cause for the problem is identified at the site and that remedial measures be taken to avoid such failures.

It is clear that the temperature of the economiser is suddenly dropping from the water wall to the distance of $x = 50$ mm, and from there, it increases to the wall on the right side. This results in low heat transfer, thermal fatigue, and tube microstructural changes. The velocity on the economiser's left-hand side (LHS) is a more right-hand side (RHS) and becomes extremely low at the economiser's mid zone from the downside. The force outage problem in the power plant is mainly due to tube leakages, and the significant contribution to the failure is erosion (fly ash erosion is more). The velocity of the flue gases is higher at every entry, so the erosion caused due to the abrasive ash is higher, resulting in more erosion rate. The CFD analysis result shows that the flue gas concentration near the economiser coil's U-bends is more elevated and hence has more chance of failure. The other reason for the forced outage is the load condition. Power plants are ready to sustain the full load conditions for a long time, so we should optimise the plant loading condition to avoid future failure.

4. Conclusion

A critical study of tube leakage data shows that the economiser tube failure is the highest in percentage due to erosion and overheating and should be addressed. CFD analysis can be a modern tool for predicting the areas and causes of tube failure in the economiser section. In the present work, the steady-state CFD model of the finned tube economiser is developed and analysed for the effect of flue gases over the finned tube economiser. The velocity, temperature, and pressure variation in the economiser were calculated using CFD simulation. The simulated velocity result was compared with the CAVT results and found validated. Further investigation of CFD results shows that the tube failure occurs at U-bends due to ash particles' high velocity and temperature. In the close view of the finned tube, we observed low temperatures at the corners of the attached fin. Most tube failures occur in this region due to significant variations in the temperatures at the welded corner, which were further validated through the tube failure data. Pressure variations show the low pressure on the downside of the tube, resulting in the un-uniform pressure of the fluid particles in close vicinity

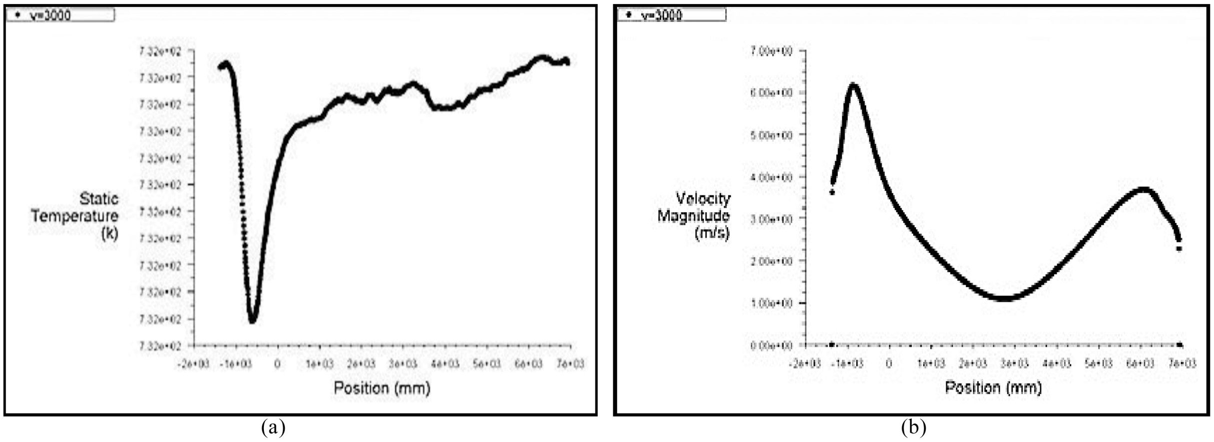


Figure 8. (a) Variation of static temperature with the position from iso-line at y = 3000 mm; (b) Variation of the static pressure of flue gases along a tube of the economiser.

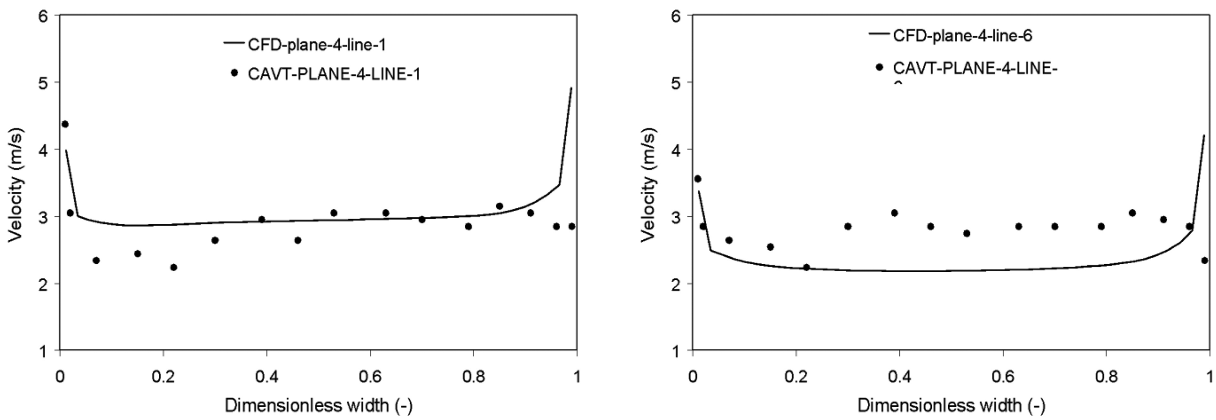


Figure 9. Comparison of predicted and CAVT results for economiser.

to the tubes of the economiser and a reduction in the heat transferring area. Redesigning the fin structure over the tube is suggested to eliminate this problem. High static temperature and velocity were observed near the walls. This helps to identify the location and magnitude of the temperature and velocity for further modification. The modification in terms of re-designing the fins over the tube, placement of the baffles in the flow passage, and introduction of the mixing chamber and blowers were suggested based on the study. In the future, transient analysis using running plant data can predict the effect of flue gases over the tube, and corrective action in process parameters can be suggested to avoid sudden failure. The impact of the flue gases can be predicted using the advanced technique of life-placing baffles, which will expect the tube life based on the simulation model and can reduce the cost of experimentation. Similar models can be built in regions like superheaters and preheaters to predict the reason for failure.

Acknowledgement

Thanks to the officers and management of Maharashtra State Power Generation Company Limited for allowing us to be a part of their 'Boiler Tube Leakage' research project and for providing me with the information and data needed for my academic achievement.

References

[1] *Monthly statistic report*. (New Delhi: Deputy Director Ministry of Coal, Government of India, Dec. 2022).

[2] *Review of performance of thermal power station 2016–17*. (New Delhi: Government of India Ministry of Power Central Electricity Authority, 2016).

[3] S. Shahul Hamid, D. Najumnissa Jamal, and M.S. Murshitha Shajahan, Automatic detection and analysis of boiler tube leakage system, *International Journal of Computer Application*, *84*(16), 2013, 19–23, doi: 10.5120/14660-2933.

[4] N.S. Drastiwati, H. Zakiyya, F.Y. Utama, and W.D. Kurniawan, Preliminary study of failure analysis on tube material boiler based on pressure aspect, *Proc. IOP Conf. Series: Materials Science and Engineering*, *288*. Bandung, 2018, 12110, doi: 10.1088/1757-899X/288/1/012110.

[5] H.A. Mrope, Y. Abeid, C. Jande, and T.T. Kivevele, A review on computational fluid dynamics applications in the design and optimization of crossflow hydro turbines, *Journal of Renewable Ene*rgy, *2021*, 2021, 5570848.

[6] A.N. Ingale, V.C. Pathade, and V.H. Tatwawadi, CFD analysis of superheater given boiler tube leakage, *International Journal of Engineering and Innovative Technology*, *1*(3), 2012, 29–31.

[7] V.T. Sathyanathan and K.P. Mohammad, Prediction of unburnt carbon in tangentially fired boiler using Indian coals, *Fuel*, *83*(16), 2004, 2217–2227.

[8] C.A. Erickson, Coal pipe erosion predictions using two phase

flow CFD, *Proc. American Society of Mechanical Engineers, Pressure Vessels and Piping Division*, San Diego, CA, 1998, 263–270.

[9] P.J. Stopford, Recent applications of CFD modelling in the power generation and combustion industries, *Applied Mathematical Modelling*, 26(2), 2002, 351–374.

[10] A. Mann, T. Dixon, F. Plaza, and J. Joyce, Opportunities for improving the performance and reducing the costs of bagasse-fired boilers, *Proc. Int. Society of Sugar Cane Technologists of the XXV Congress*, Antigua Guatemala, GT, 2005, 241–247.

[11] G.K. Gupta and S. Chattopadhyaya, Critical failure analysis of superheater tubes of coal-based boiler, *Strojniski Vestnik/Journal of Mechanical Engineering*, *63*(5), 2017, 287–299, doi: 10.5545/sv-jme.2016.4188.

[12] E.D.T. Durga Prasad, K. Kiran Kumar, and A. Santosh, *CFD analysis on various finned tube heat exchanger*; Ph.D. Thesis, Andhra University, Visakhapatnam, India, 2018.

[13] T.C. Mohankumar and Thomachan, CFD studies on multi lead rifled [MLR] boiler tubes, *International Journal of Engineering Research and Applications*, *3*(5), 2013, 24–26.

[14] P.A. Bendekar, P.M. Kumar, and P.S. Poojari, Analysis of economizer using the CFD tool, *Proc. 3rd Int. Conf. on Recent Trends in Engineering Science and Management*, Bundi, 2016, 694–700.

[15] K. Sainath, M. Salahuddin, M. Shafi, and T.K.K. Reddy, Comparative study of economiser using the CFD analysis, *International Journal of Modern Engineering Research*, *4*(4), 2014, 1–6.

[16] C.R. Babu, P. Kumar, S. Roy, and D. Kanungo, CFD analysis of an economizer for heat transfer enhancement using serrated finned tube equipped with variable fin segments, *Materialstoday Proceeding*, *45*, 2020, 222–230, doi: 10.1016/j.matpr.2020.10.422.

[17] A. Thirupathaiah, Modelling and computational fluid dynamics analysis of economizer in tangential fired tube boiler, *International Journal of Innovative Technology and Research*, *6*(6), 2018, 8886–8890.

[18] K.P. Mudafale and H.S. Farkade, CFD analysis of economizer in a tengential fired boiler, *International Journal of Mechanical and Industrial Engineering*, *3*(2), 2013, 143–147, doi: 10.47893/ijmie.2013.1144.

Biographies

Chandrakishor L. Ladekar received the M.Tech. degree in thermal engineering from GCOE Amravati in 2008 and the Ph.D. degree in mechanical engineering from RTMNU in 2019. He has 21 years of teaching experience and currently, he is an Associate Professor of Mechanical Engineering with PCCoE, affiliated SPPU Pune. His research area includes analysis of energy and thermal systems, BTL, and heat pipe.

Manoj R. Dahake received the M.Tech. degree in thermal engineering from GCOE Amravati in 2008 and the Ph.D. degree in mechanical engineering from SPPU in 2022. He has 15 years of teaching experience and currently, he is an Assistant Professor of Mechanical Engineering with AISSMS CoE affiliated SPPU Pune. His research area includes heat transfer analysis, IC engine simulation, and fuels.

Chandrakant B. Kothare received the M.Tech. degree in thermal engineering from GCOE Amravati in 2008 and the Ph.D. degree in mechanical engineering from RTMNU in 2020. He has 21 years of teaching experience and currently, he is working as an Associate Professor in Mechanical Engineering with SSACE Wardha affiliated RTMNU, Nagpur. His research area includes CFD, heat transfer, IC engine, and fuels.

Ajay V. Kolhe received the M.Tech. degree in heat power from VNIT Nagpur in 2007 and the Ph.D. degree in mechanical engineering from RTMNU, Nagpur, in 2015. He has 23 years of teaching experience and currently, he is an Associate Professor of Mechanical Engineering with KITS, Ramtek affiliated RTMNU, Nagpur. His research area includes CFD, combustion analysis, and turbulence model flow.