A LITERATURE REVIEW ON COMPUTATIONAL FLUID DYNAMIC ANALYSIS OF WATER TUBE BOILER AND HEAT EXCHANGER TECHNIQUE

SHOBHIT KUMAR GAUTAM[1], Dr. R.K.MANDLOI (PROFESSOR MANIT BHOPAL)[2]

---***---

ABSTRACT: The objective of this paper is to investigate the heat transfer flow rate inside the water tube boiler in different geometries like corrugated tube shapes, multi lead rifled tubes, internally grooved tubes etc. In this paper we review's the problem related "how to enhance the heat transfer and performance of water tube boiler by using CFD tool". And to reduce the maintenance cost of the boiler tube. Various researches have been done to improve the conditions of water tube boiler. Previously research used the different geometry's of boiler tube, like using of multi lead rifled tube or varying corrugated tube shapes, instead of plane internal surface of the boiler tube to increase the heat transfer and performance. the use of different geometries with riffles and corrugated plays a vital role in fluid flow phenomena as the internal surface of boiler tubes are planed walled so the nature of fluid flow is laminar which can't maximize the heat transfer. due to use of riffles inside the tubes creates the turbulence in the flow which convert the nature of fluid flow from laminar to turbulence flow due to turbulence flow inside the tube, the heat transfer rate increases so the use of internally grooved or riffles inside the tube surface gives the better results. This paper provides the detailed suggestions of geometry of water tube boiler which can enhance the performance and efficiency of the boiler.

Keywords- computational fluid dynamic [CFD], multi lead rifled [MLR], geometrical parameters.

INTRODUCTION:

The major source in industries for the combustion of fuel is boiler which generates electric power and steam. Steam generation is the principal device. The increase in energy costs in recent years necessitates the need to use more efficient systems; engineers try continuously to increase the efficiency of steam production rate. Thus the design of boiler tube geometry is an important criterion for enhancing heat transfer the influence of geometric parameters depends on operating conditions such as operating pressure , temperature, mass flow rate, contact surface etc.

In actual boiler tubes the internal surface are plane wall tube, due to which the nature of flowing fluid inside it are laminar, the use of smooth surfaces inside the tubes result in poor performance, it reduces the heat transfer rate and also scale formation on the internal wall of the tube, due to which life of boiler tubes decreases and the maintenance cost increases. To overcome this problem we need to increase the contact surface area and create the turbulence inside the tube so that heat transfer rate increases and scale formation reduces. A new geometry of the boiler tubes should be design and analyze with the help of CFD tool. The use of CFD software is very useful in this research work as it reduces the cost of experimental set–up as well as it will save time also. CFD has become a important tool in the industry cycle not just for researching and Designing new systems but also recognizing current ones and developing them. Except in complicated geometries, forecasts of drop in pressure and heat transfer are always accurate. Thus CFD has become the state of the art in engineering technology as in the development of heat exchangers. As in power plants the boiler tubes are made of steel which has thermal conductivity around 50.2W/mk if we replace the boiler tubes material with cooper which has thermal conductivity around 385W/mk then the heat transfer also be increased but copper tubes are costly then steel so we can't use it.

Some of the advantages of Computational fluid dynamics:

Reduction of production costs: It can be costly to use practical trials and testing to get critical design engineering data. CFD simulations are relatively inexpensive, and costs are projected to decrease as systems are more powerful. Rapid evaluation of the feature differences: CFD tests can be carried out in a limited time. In the planning process, technical details should be brought in early. Thorough data: Experiments just grant the extraction of information at few gadget destinations (where indicators and strain checks are introduced). CFD enables the intending to assess each position in the zone of intrigue and view its result through an assortment of warmth and stream factors. Requires the purchaser to reproduce various conditions, and often stream and thermal performance systems cannot be easily checked. CFD is capable of modeling any physical state possibly; CFD has great control over the physical area, providing the opportunity to differentiate different test parameters.

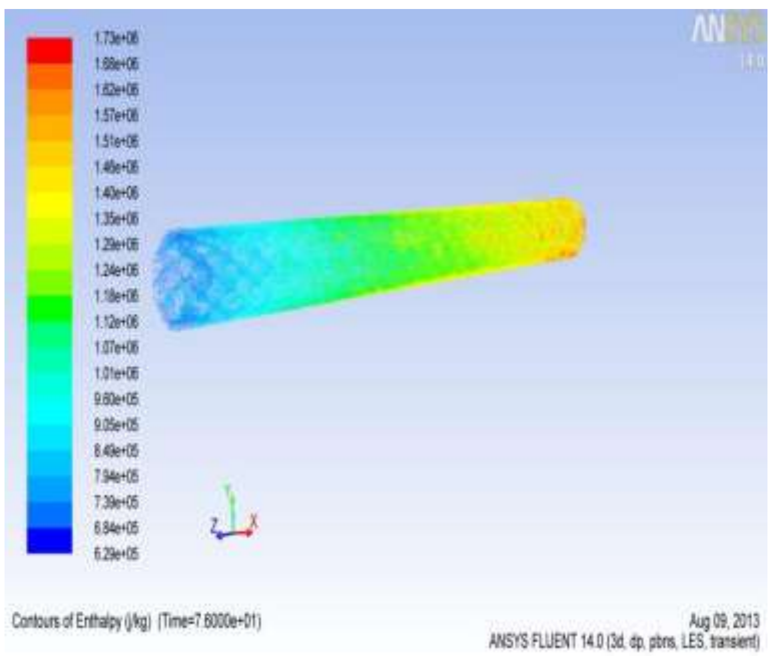
Literature survey

1. CFD Studies on Multi Lead Rifled [MLR] Boiler Tubes

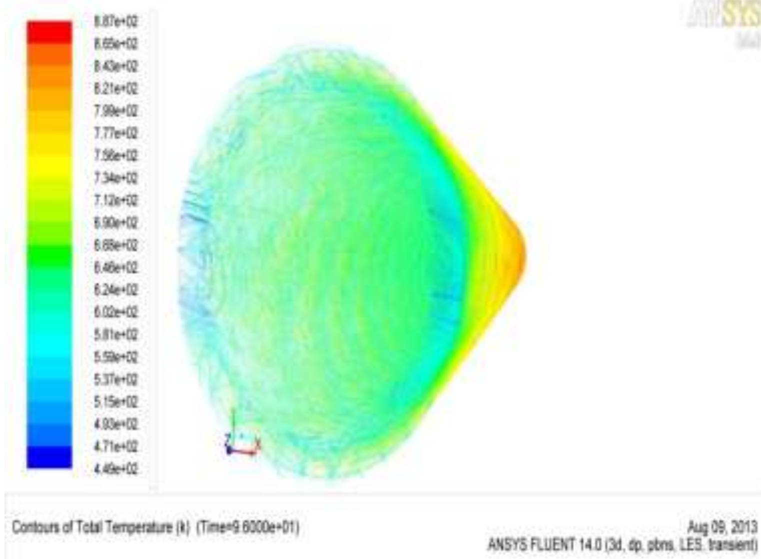
Investigations(by Dr. T C Mohan kumar, Nice Thomachan) shows CFD tests on multi-lead rifled[MLR] boiler tubes were conducted to evaluate the tube flow using Ansys14 and Geometry based on solid works 2012 was moulded using Gambit2.4.6 and Steel and water material properties were copied from data from the fluent ansys14 program. He created a quadrilateral mesh uniformly across the area and analyzed using fluent ansys package 14.5.

By using the CFD simulation it will result in Better heat flow according to the boiler tube surface region and several other rifling results. In actual situations, Improved heat transfer area and increased water contact to the tube wall are crucial changes in heat transfer. For certain areas, Steam can be in tube wall touch, steam heat absorption efficiency is poor compared to water, So that there is no adequate cooling of the tube in that region.

A swirling flow in the MLR tubes and a centrifugal force separating steam and water, By holding less concentrated steam in the center field, it takes denser water to the wall sector. This influence of the MLR boiler tube makes heat transfer more effective than the normal boiler tube for the plane. The boiler performance Will indicate heat transfer enhancements in boiler furnace. A slight rise in the tube's heat transfer will bring huge change to the boiler's overall performance. The outcome of this paper is that the boiler being studied has 332 Vertical tubes arranged from the top of the boiler to its bed coil (steam drum), the program analyzes the sample length and displays the resulting temperature, enthalpy contours and plots are given below:



Contours of Enthalpy



Contours of Temperature

2. CFD ANALYSIS OF HEAT TRANSFER ENHANCEMENT BY USING PASSIVE TECHNIQUE IN HEAT EXCHANGER

Investigation by C Rajesh Babu and Santhosh Kumar Gugulothu is Assistant Professor, of Department of Mechanical Engineering, GITAM University,HYDERABAD, INDIA

In this paper they talk about CFD tool is however very important and effective tool to understand heat transfer applications. Modeling the heat transfer flow of computers was one of the big problems facing classical science Nonetheless, CFD is a very important and powerful method for understanding heat transfer applications Modeling the numerical analysis, it is a major challenge in the classical sciences is heat and mass transfer flow. The heat transfer enhancement is improved by the inclusion of the inserts due to their significance in various applications. Heat exchanger is a device designed to effectively move heat from one medium to another. We require heat exchange between two fluids at separate temperatures so that they do not mix with each other. Specific heat exchanger implementations include Condensers, evaporators, heating and refrigeration boilers etc. In this paper they analysis the heat exchanger by a CFD software to improve the heat transfer with the help of a passive technique. The Strengthening heat transfer is very important in many engineering applications to increase heat exchanger performance. The active techniques required outside power, such as surface vibrations, Electrical fields etc. and passive strategies are those where no external control is needed however inserts are needed to interrupt the flow like tape inserts etc. However, literature survey says that passive techniques provide more heat transfer capacity without the need for external power by keeping specific tape inserts. CFD technology is therefore very necessary and efficient method to consider heat transfer applications. Simulating heat transfer flows by computer is one of the main challenges in the classical sciences. The heat transfer enhancement is improved by the inclusion of the inserts due to their value in various applications. By modeling CFD by taking condensed tube and considering heat transfer enhancement with and without inserts, we conclude using ANSYS Fluent version 14.5.

By using different techniques we come to the conclusion that the transition of heat Coefficient increases with reduced pressure various heat Improvement techniques are sorted as follows:

Passive Techniques:

Passive techniques are geometrical changes, or by keeping inserts to disturb the fluid. They promote higher heat transfer coefficients by disturbing or altering current flow behavior (except extended surfaces) which also leads to higher pressure drop. Heat transfer augmentation achieved by following:

2.1.1Treated Surfaces: The treated surfaces are primarily applicable in heat transfer in two phases, and Consists of different formed surfaces (continuous or discontinuous integral roughness of the
surface Or alterations) and coatings. Where this procedure gives the surface a "roughness," Its size (normal surface protrusion) is not sufficiently high to affect forced single-phase Convectivity.

2.1.2 Rough surfaces:

Structured roughness may be intrinsic to the surface, or wire-coil-type attachments can be used to create the protuberances. The former will be rendered by machining, cutting, casting or welding (e.g. knurling, threading, grooving). and the resulting surface area Their geometrical structure can contain two-dimensional or distinct three-dimensional proturberances or grooves. These are some techniques which are used to improve heat transfer and
The methodology used for performance assessment using CFD analysis is:

- Domain Flow Identification
- Designing the Structure.
- Produced system.
- Boundary specified Conditions.
- Solver feature collection and
- Conditions for convergence;
- Tests and post-treatment.

After following this methodology the result of this paper is:
CFD analysis is done through the use of dual pipe heat exchangers with cold and hot fluids with various boundary conditions with the application of helical tape inserts. This can be inferred as follows: by using passive methods , i.e. by inserting helical tape, heat transfer inserts Improvement increased by 10-15% at the expense of a fair permissible pressure drop Through this article, we have achieved an effective improvement in heat transfer.
Potential studies can be applied to the following:
Applications that have high thermal conductivity applications should be converted to copper from aluminum. Combination of techniques may be used to enhancement of heat transfer coefficient by compound techniques.

3. **Study and Analysis on Boiler Tubes for Performance Enhancement with Varying Corrugated Tube Shapes**

Vimal Singh Chamyal1 Bhavana Singh1, Shivasheesh Kaushik2 Sanjay Kumar1 Mohit Pant1 and Tarun Tripathi1 are investigating this paper, they are the Research Scholar, Mechanical Engineering Department, Amrapali Group of Institute, Haldwani, Uttarakhand, India,

This research paper uses a commercial computational fluid dynamics (CFD) method for connective heat transfer across grooved or folded channels. Improvement of heat transfer by the corrugated tubes has been studied, by changing the geometrical dimensions of the corrugated tubes. Including rifling, height if rifling, pitch length of rifling, etc.it is easy to study the performance of geometry of corrugated tubes. Flows through the tube of heat exchanger were determined by the effects of different geometry of the corrugated tube such as triangular elliptical, hexagonal etc. The change in the shape of corrugated tubes gave the significant change in heat transfer. The result shows that heat transfer rate increases as compared to the water tube's inner plane surface.
Following methodology is followed in this paper to obtain the result,

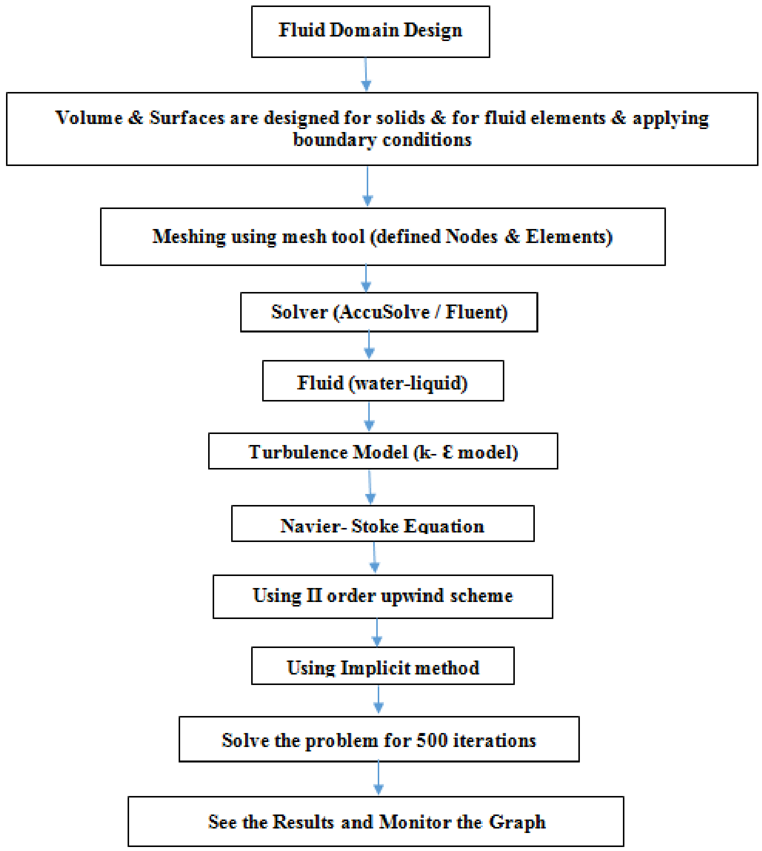
3.1 MATERIAL AND METHEDOLOGY

The present research is split into four phases-

- Modeling geometry
- Pre trieval
- Recording
- Post-treatment

The current research method can be divided in Four phase flow phases, which are simulation of geometry; Pre-processing, refining and post-processing.

3.2 This flow chart is used to solve the problem



These dimensions are used to design rifled tube geometry and parameters are:

- Tube type **Helical ridging**
- Outer diameter (mm) - 5.08 cm
- Helix angle - 30o
- Length - 150 cm
- Tube material - Steel
- Length of pitch - 25 cm
- Height of rifling - 0.11 cm

In general, the behavior of the flow is governed by the fundamental principles of the classical mechanics expressing conservations of mass and energy, following assumptions is to be taken to analysis the models which are:

Assumptions are:

1. Flow must be steady.

2. Flow must be incompressible.

3. Flow should be turbulent.

Type	Inlet	Outlet
Total Heat Transfer Rate (W)	422.101	3381.392
Temperature (K)	300	391
Coefficient of heat transfer (W/m2-K)	-	2636.6

This model is studied; resolved and validated by Usage of the ANSYS (FLUENT) programmed, In the present work, the boiler's parametric study of tube geometry and the heat transfer characteristics of the boiler tube are different in geometry and compared with the boiler tube Smooth tube. Result of this paper is given below:

For this study, the heat transfer factor and its intensity are analyzed. A slight increase in the Coefficient for transfer of heat will result in a massive Increase entirely boiler performance depending on the geometry of the boiler tubes, In the software the sample length of the boiler tubes was studied, Contours, plots and resulting temperature enthalpy. The turbulence was caused by corrugated boiler tube channels Fluid that gives higher rate of heat transfer.

4. **CFD analysis of an innovative heat recovery system :**

Robert, Andrei Burlacua, Dorina Nicolina Isopescua, Marina Verdea, Gavril Sosoia, Constantin Doru Lăzărescua, written this article, This paper discusses a heat transfer study of the computational fluid dynamics (CFD) for an initially developed method for energy recovery from excess hot water in the energy sector. The goal of this research is to improve the quality of a heat transfer of heat pipes utilizing phase shifting materials. The tool can reduce a building's energy needs So, it can be a safe alternative thermodynamic energy source. The continued degradation of the atmosphere and the rising fuel scarcity problems have contributed to a greater need for renewable energy sources. As the building industry consumes about 25-45 percent of the overall amount of energy generated in the developing worlds, buildings' energy efficiency has become an widely debated topic worldwide. Several analysis and simulations of various parameters were conducted for each scenario to achieve the purpose of this paper. The heat exchanger geometry was designed in 3D with the aid of Autodesk Inventor Technology. To evaluate the energy efficiency of the proposed device. The CFD study was conducted using Autodesk CFD modeling software. The study shows that the heat exchange approach can be a viable choice for enhanced conserving energy in residences. The process will restore high amounts of heat energy from remaining sources at low temperatures which can be used to warm or heat up the hot water in a built environment. expanding the secondary agent's mass flow rate will minimize the total heat transfer capacity by moving away the circulation in a defined position, The extracted heat energy is growing Paraffin-coated metal pipes reflect a fair boost for the heat exchanger 's efficiency, When they pass over small gradients of temperature. Paraffin's ability to conserve thermal energy is also an important advantage due to the increasing intermittent heat source system, operating and maintenance costs are relatively low, as the machine is a sensitive device that does not have its own filtering system. The PCMHE (Phase Change Material Heat Exchanger) can minimize a building's thermal needs while reducing CO2 emissions and can also have an economic effect.

5. Enhanced thermo-hydraulic performance in a V-ribbed triangular duct solar air heater: CFD and exergy analysis:

In this paper they conducted a Computational Fluid Dynamics (CFD) as well as energy and exergy to explore the effect of secondary flow provided by V-ribs on the overall effectiveness of a triangular solar air heater (SAH) duct. The effect of rib inclination (α) is measured for a given relative rib pitch (Rp= 10) and relative rib height (Rh = 0.05) using CFD technique to modify the number of Reynolds (5000 / Re 20000). Based on the CFD simulation results, empirical relationships are established which can predict Nu and f with a relative deviation of 8.7% and 4.7% respectively. Using those correlations, performance analysis is exercised. The average efficacy parameter (π) of 2,01 for α = 45 ® set is obtained at Re ≤ 7500. The energy analysis shows that the entropy generated for the triangle corrugated duct is lower than for the smooth duct with maximum exertion efficiency increase as 23 percent for α = 45. The analysis will be expanded to equate the efficiency of the rectangular duct with the triangular duct corrugated with SAH (α= 45®). Tests demonstrate that in various configurations, the ribbed triangular SAH duct (α= 45 ®) is superior Higher mass flow levels of the corrugated rectangular conduct SAH (solar air heater).

6. CFD analysis of the performance of elbow-meter with high concentration coal ash slurries:

In this paper they conduct an elbow meter CFD study, Elbow meter is a direct path measuring instrument and its fluid dynamic properties are well understood, and the elbow meter coefficients(Ck) depending on parameters such as Reynolds number, diameter ratio, friction of the pipe etc. There is a well-known literature of, Elbow meters are often used to filter liquid solids for other industries. The present research aims at checking the characteristics of an elbow meter using approved CFD for high concentration coal ash sludge pipelines Large concentration of coal ash sludges is known to behave as relatively homogenous fluids that exhibit behavior as liquid fluids are liquid Bingham. The developed CFD technique was used to predict Ck values for the flowing of plastics fluid from Bingham and to determine its radius, hedstrom number and Bingham Reynolds number dependence. In addition, Ck is found to be independently of Hedstrom Number for any specified radius ratio for the high concentration fly ash slurry flows (which was investigated across the period He ≤105). In fact, Ck stays stable in fully turbulent flows, over a critical number of Reynolds ($Re \geq 5.3 \times 10^3$) and depending on the radius ratio only.

7. Thermal performance analysis and experimental validation of primary chamber of plasma pyrolysis system during preheating stage using CFD analysis in ANSYS CFX

In this paper they performed a CFD analysis in ANSYS CFX of a plasma pyrolysis system, Plasma pyrolysis arises globally as a very efficient method to proper handling of medical waste. The plasma arc plays a crucial role in achieving the necessary extreme temperatures of nearly 1000 ° C in the reactor vessel until the wastes are poured into such a cavity and the garbage is decayed with no toxic substances or through-products. Simulation of the Transient CFD (Computational Fluid Dynamics) To evaluate the temperature and its relation with the device, Usage of the widely available CFD process ANSYS CFX. Apparently only a pre - heating study has been done without pollution or other chemical reactions being added. It would help with the construction of a high-potential plasma pyrolysis system (200 kg/hr. or more) by justifying the methods used in this study and its conclusions. During the primary chamber preheating level, heat loss, energy consumption etc. For most of the experimental period, CFD results were found to be less than a comparable range (< 20 per cent for the internal layer and < 10 per cent for the outermost layer). The paper also explained the challenges for that divergence.

8. Thermodynamic performance of a CO_2 vortex tube based on 3D CFD flow analysis:

In this paper they performed a CFD flow analysis on the vortex tube (VT). The Vortex tube (VT) is a mechanical system designed to include heating systems simultaneously. In this research a 3D CFD analysis is first developed to replicate the CO2 flow within a flow field and is then tested using experimental evidence recorded. The assumed k-€ turbulence model makes use of organized The hexahedral modules produced in ANSYS mesh generation. The established CFD model is paired with the VT (vortex tube) thermodynamic model to complete a parametric analysis, where the parameters chosen are the air intake intensity (500 KPa to 1350 KPa) and the cool molar ratio (0.2to0.9) respectively. This shows the effect on the isolation and efficiency of VT energy, The energy separation is described in terms of the variations between both the hot exit and the cold exit temperature, in both relation to the VT inlet temperature Quality is determined by measurements of freezing strength, warming ability and energy and exertion. Statistical findings indicate that the variance of the cold volume fractions from 0.2 to 0.9 makes a gap in hot exit temperature to increase from 10.2° C to 78.7° C for a given inlet intensity of 1350 kPa, while the temperature differ for the cold exit falls from 44 °C to 9° C. Throughout this test, CFD work was performed to show energy isolation Throughout the CO2 vortex-tube as a true difference in the common inlet intensity and the cold volume fraction. Separate Power as for the variations between hot outlet and cold outlet temperature was addressed. All temperature of the VT inlet Output is defined by freezing, warming, Heat and exergy measures of increased inlet pressure, temperature adjustments for the cool outlet and elevated hot outlet temperatures.

Conclusion:

Computational fluid dynamics (CFD) is a research discipline Dedicated to the simulation of fluid flow, heat transport, heat flow, chemical processes and related parameters, utilizing a computational method to solve mathematical models regulating such phenomena. Through utilizing this technique, manufacturers may check that their goods follow the requirements of a consumer early on in the design process, In these papers we observed that heat transfer in water tube boiler is depend on surface area of tubes, mass flow rate of fluid, geometry of tubes, material of the tubes and nature of fluid flow, so in these paper they analysis on circular tube triangle tube and corrugated or riffled tubes with the help of CFD software which gives a great result in heat transfer. In this analysis some more work can be done on different geometry like elliptical tubes, hexagonal tubes etc., and we can change the material of tube instead of steel we can used the material which has higher value of heat transfer coefficient like copper, so that overall heat transfer improves and which leads to increase in efficiency of boiler as well as increase the output of the plant, adding the riffles in the geometries play's an important role due to riffles nature of the fluid flow changes from laminar to turbulent flow which increases the heat transfer, so for better results we can design a new geometry like elliptical, hexagonal tubes with riffles.

Acknowledgement: It gives me immense pleasure to express my deepest sense of gratitude and sincere thanks to my highly respected and esteemed supervisor Dr. R.K.Mandloi (Professor, Department of Mechanical Engineering) for his valuable guidance, encouragement and help for completing this review paper.

References:

1. Mohankumar, T. C., Thomachan, N., Engg, M. & College, M. E. S. OPEN ACCESS CFD Studies on Multi Lead Rifled [MLR] Boiler Tubes. **3**, 24–26 (2013).

2. Wang, H., Zhang, C. & Liu, X. Heat transfer calculation methods in three-dimensional CFD model for pulverized coal-fired boilers. Appl. Therm. Eng. **166**, 114633 (2020).

3. Selvaraj, P., Sarangan, J. & Suresh, S. Computational fluid dynamics analysis on heat transfer and friction factor characteristics of a turbulent flow for internally grooved tubes. Therm. Sci. **17**, 1125–1137 (2013).

4. Vizitiu, R. S. et al. CFD analysis of an innovative heat recovery system. Procedia Manuf. **32**, 488–495 (2019).

5. Granda, M., Trojan, M. & Taler, D. CFD analysis of steam superheater operation in steady and transient state. Energy **199**, 117423 (2020).

6. Aghagoli, A. & Sorin, M. Thermodynamic performance of a CO2 vortex tube based on 3D CFD flow analysis. Int. J. Refrig. **108**, 124–137 (2019).

7. Sharma, D. et al. Thermal performance analysis and experimental validation of primary chamber of plasma pyrolysis system during preheating stage using CFD analysis in ANSYS CFX. Therm. Sci. Eng. Prog. **18**, 100525 (2020).

8. Rawat, A., Singh, S. N. & Seshadri, V. CFD analysis of the performance of elbow-meter with high concentration coal ash slurries. Flow Meas. Instrum. **72**, 101724 (2020).

9. Pini, A., Cammi, A., Lorenzi, S., Cauzzi, M. T. & Luzzi, L. A CFD-based simulation tool for the stability analysis of natural circulation systems. Prog. Nucl. Energy **117**, 103093 (2019).

10. Nidhul, K., Kumar, S., Yadav, A. K. & Anish, S. Enhanced thermo-hydraulic performance in a V-ribbed triangular duct solar air heater: CFD and exergy analysis. Energy **200**, 117448 (2020).

11. Ghafori, H. Computational fluid dynamics (CFD) analysis of pipeline in the food pellets cooling system. J. Stored Prod. Res. **87**, 101581 (2020).