Analysis of Shell and Tube Heat Exchanger for Waste Heat Recovery by using CFD

[1]Dr.D. Santha Rao,[2]Arbaaz Khan,[3]Subhasish Modak,[4]Bani Venkatesh,[5]Salina Pavan

[1]Professor,[2,3,4,5] UG Students

[1]Department of Mechanical engineering,

[1]Godavari Institute of Engineering and Technology(A), Rajahmundry, Andhra Pradesh, INDIA.

Abstract: *The heat available in the exhaust section of many energy conversion device such Boiler, Gas turbine, I.C engine etc goes as waste, if properly not utilized. Currently the work has been carried out with a view to assume the performance of a shell and tube heat exchanger in the recovery of waste heat application. The performance of the heat exchanger has been evaluated by using the Computational fluid dynamics package in ANSYS fluent 15.0 and the parameter of performance of the heat exchangers are given in effectiveness, waste heat recovery, logarithmic mean temperature distribution has been reported in this report.*

Key-words- Recovery of waste heat application, Computational fluid dynamics.

I. INTRODUCTION

Energy is a very essential entity for any economic development of any country in this globe [8]. At the current scenario there is a tremendous rapid industrialization growth, which is the main relevant reason for the energy crisis and also for pollution which is causing global warming. As the current scenario, we are widely using Diesel Engine widely used in many industries. Almost 2/3rd of the energy is wasted by the exhaust gas which is directly or indirectly causing global warming overall energy requirement [8]. Depending on the temperature level of the exhaust stream and proposed application, different heat exchanger devices, heat pipes are used combustion equipments has been working to utilize the use of recovered heat. Before, Shell and heat tube exchanger was broadly used as industrial heat transfer equipments. Plain tubes were used during those periods for assembling of the fins. After industrialization, now a day's modified tubes are used for proper exchanging of heat just like the finned tube. Heat transfer coefficient selected for the experimentation are is demineralized water which is higher than gases, which also reduce the size of the heat exchanger.

Demineralized water is mainly used in the purpose of ion exchange, electro deionization, or membrane filtration technologies, which can be more useful for creating an ultrapure water process such as distillation.

Demineralization by ion exchange, electrode ionization or membrane filtration can produce water that is nearly 100% free of minerals and salt including Alkalinity, Calcium, Chloride, Iron, Magnesium, Nitrate, Potassium, Silica, Sodium, Sulphate. Demineralized water can be effective for many things such as Power, Refiner, Petro-chemical and In this section efforts on works done by past researchers on Shell and Tube Heat Exchanger CFD analysis have been discussed.

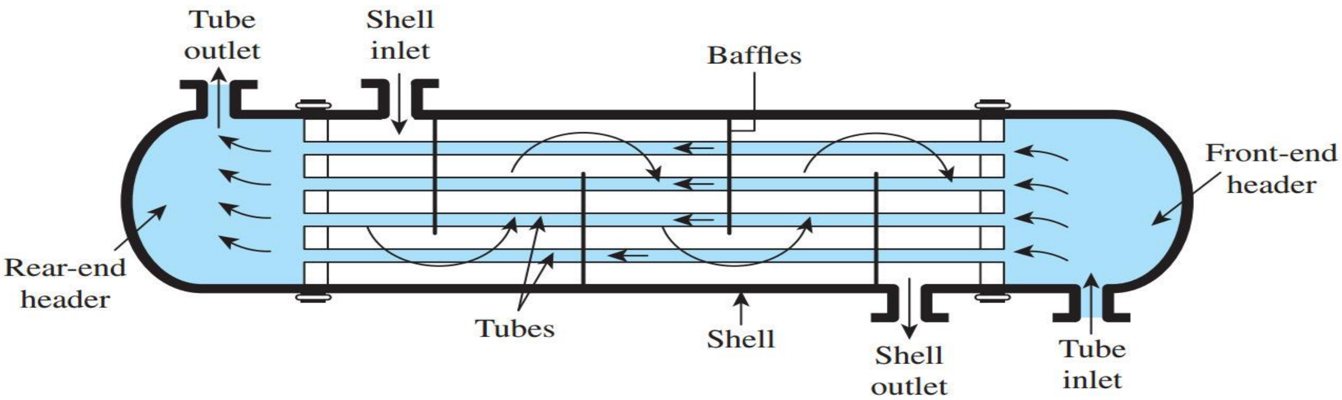


Fig.1 shell and tube heat exchanger

II. ROLE OF HEAT EXCHANGERS

A heat exchanger is a framework used to transfer heat between at least two fluids. Heat exchangers are utilized in both cooling and warming procedures. The liquids might be in direct contact or might be isolated by a strong divider to forestall blending. They are generally utilized in space heating, refrigeration, cooling, power stations, synthetic plants, petrochemical plants, oil treatment facilities, gaseous petrol preparing, and sewage treatment. The extraordinary instance of a Heat exchanger is found in an IC Engine in which a revolving around fluid known as Engine coolant travels through radiator circles and wind streams past the twists, which cools the coolant and warms the moving toward air.

Another model is the warmth sink, which is an aloof heat exchanger that moves the warmth produced by an electronic or a mechanical gadget to a liquid medium, frequently air or a fluid coolant. There are three essential orders of Heat exchangers as indicated by their stream plan. In Parallel Flow heat exchangers is the type of heat exchanger in which different fluids enters in the exchanger from two different sides and traveling in relating to one another to the contrary side. In counter-Flow heat exchangers the liquids enter the exchanger from furthest edges. The counter-current plan is the most proficient, in that it can move the most warmth from the warmth (move) medium per unit mass because of the way that the normal temperature contrast along any unit length is higher. See countercurrent trade. In a cross-flow heat exchanger, the liquids venture out generally opposite to each other through the exchanger. For boosting up the efficiency, heat exchangers are intended to boost the surface territory of the divider between the two liquids, while limiting protection from liquid course through the exchanger. The exchanger's presentation can likewise be influenced by the expansion of balances or creases in one or the two headings, which increment surface zone and may channel liquid stream or actuate disturbance. The driving temperature over the warmth move surface changes with position, however a proper mean temperature can be characterized. It is based on the main theorem of "log mean temperature difference" (LMTD). Some of the time direct information on the LMTD isn't accessible and the NTU technique is utilized.

III. MODELING OF HEAT EXCHANGERS

The dimensions of the heat Exchanger's value which are majorly used in different types of industries with different parameters of design. The standard dimensions which are taken for modeling are:

Table 1.1: dimension of the shell and tube heat exchanger

SL.NO	OBJECTS	DIMENSIONS (IN MM)
1	Length of the tube	610
2	Length of the shell	610
3	Number of tubes	21
4	Number of baffles	13
5	Total Length of the STHE	810
6	Thickness of the shell	5
7	Thickness of the tube	4
8	Height of the inlet valve of both shell and inlet cap	20
9	Height of the outlet valve for both shell and outlet cap	20
10	Diameter of the shell	160
11	Diameter of the tube	20

Based on the given data in the table, heat exchanger has been modeled.

IV. METHODOLOGY WITH COMPUTATIONAL FLUID DYNAMICS:

Computational Fluid Dynamics deals with the numerical examination and information structures to break down and take care of issues that include fluid flows and also part of fluid mechanics. Personal Computers are utilized to play out the computations required to reenact the free flow of the fluid, and the communication of the fluid (fluids and gases) with surfaces characterized by limit conditions. With fast supercomputers, better arrangements can be accomplished, and are regularly required to comprehend the biggest and most complex issues. CFD is applied to a wide scope of research and building issues in numerous fields of study and businesses, including streamlined features and aviation investigation, climate recreation, common science and ecological building, mechanical framework plan and examination, natural designing, liquid streams and warmth move, and motor and ignition investigation.

Computational Fluid Dynamics process comprises of three essential advances:

1. Pre-processing
2. solution
3. Post-processing

4.1 Pre-Processing: This is the Primary step of the CFD simulation procedure which helps in describing the best possible way. In this step concerned inside the pre-processor degree is to define a suitable geometry wherein the fluid flow trouble is to be analyzed. Now design is done by using the fluid flow(fluent) in design modeler by taking the above data,

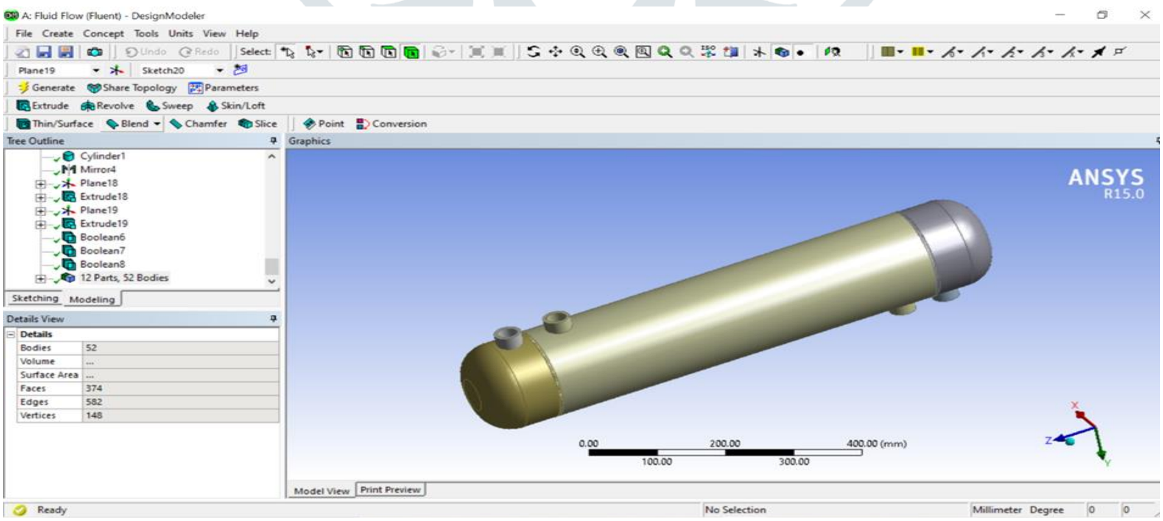


Fig.4.1.1 Geometry design of shell and tube heat exchanger

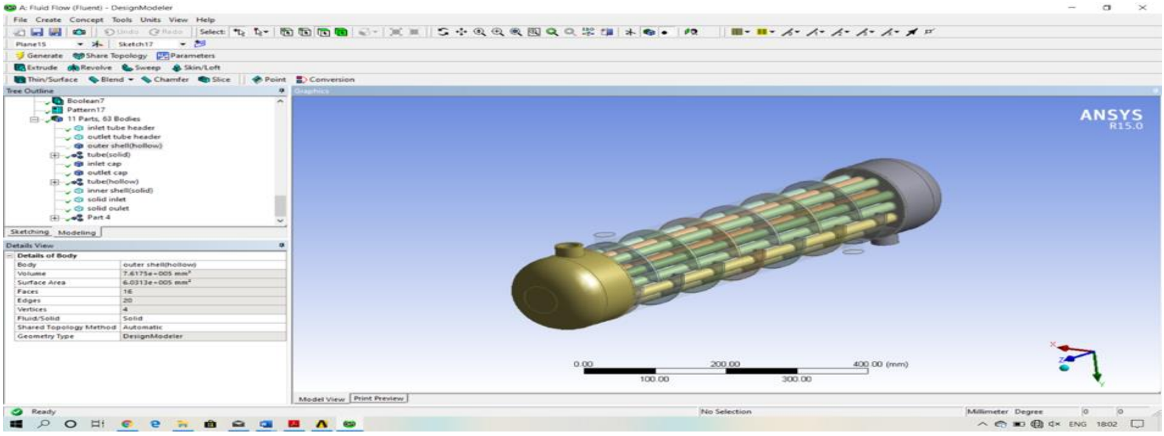
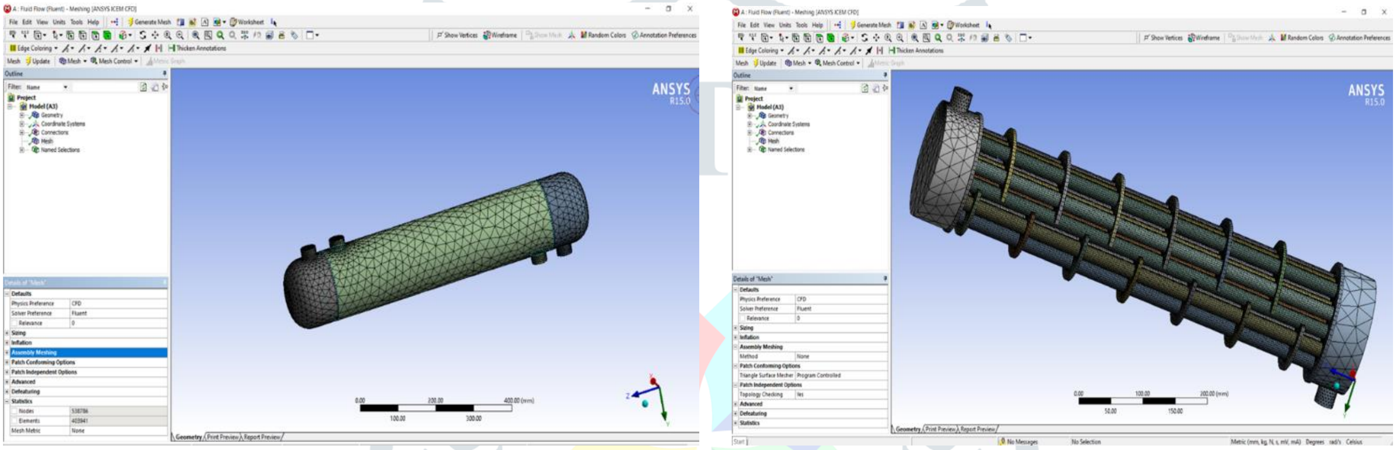


Fig.4.1.2 Inside view of shell and tube heat exchanger

Meshing of Geometry (discretization of the computational domain):

A mesh can be defined as an extent that has been discretized into a series of smaller volumes or elements. The mesh in the finite volume method considers points that form a set of volumes which are called cells. The finite element methods use sub-volumes called elements that have nodes where the variables are defined. The values of the nodes are 538786 and for the elements are 403941 obtained.



4.1.3. Mesh generation of shell and tube heat exchanger

4.2 Solution:

After meshing we added the body into solution setup condition for analyzing the results by filling some standard data. First, we have put the general conditions which are Steady state type and the unit is converted into mm. Then the second step is to model, where we are inserting the energy equation ON and K epsilon value. Then material selection dialogue appears in which copper material is selected for tube and steel material for shell. For fluid, water and air is added from the fluid database.

Cell-zone condition is defined as assemble the required parameters. Boundary-zone condition is also defined as to allocate the boundary condition for fluid flow, the shell side fluid is exhaust gas which is coming from a diesel exhaust and having the temperature of 1200C and outlet is defined as Outflow. Tube side temperature is taken as 20 and outlet temperature defined as outflow respectively. Keeping the shell side mass flow inlet as 2 Kg/s, and the tube mass flow rate is 0.5 Kg/s at the given setup and then the calculation is been run.

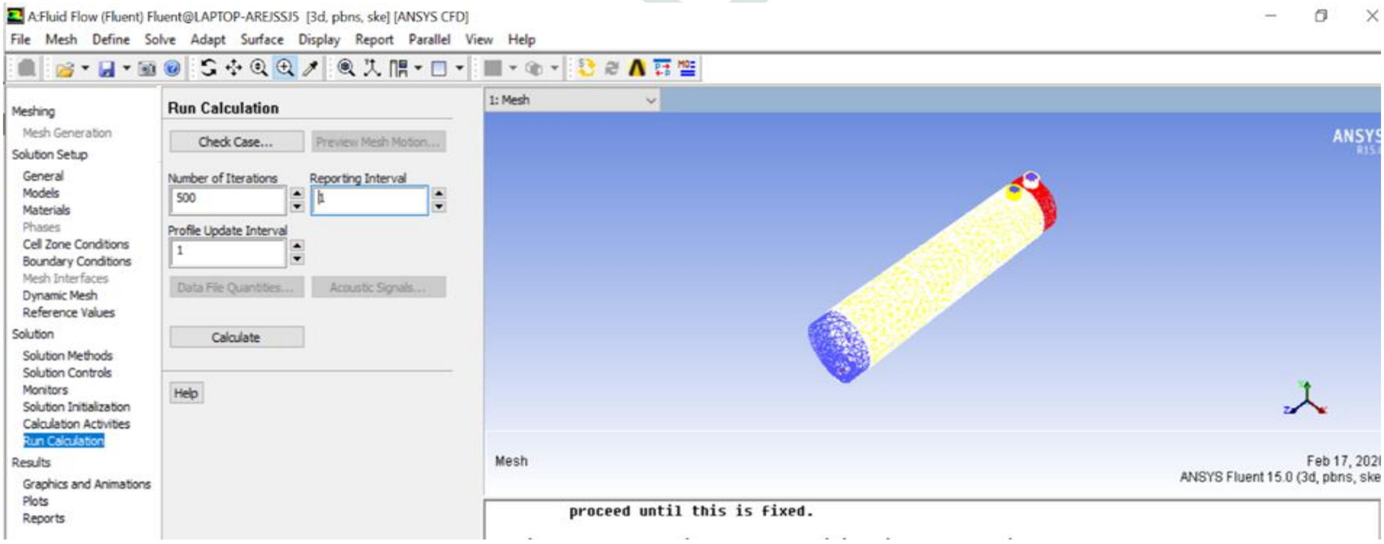


Fig.4.2.1 Setup initialization for running the calculation of STHE

4.3 Post-Processing: After the exporting the model into the CFD-post the analysis has done. In that the results in the condition of contour plot, streamline, vector plot has been defined.

So, first of all we take the y-z section at the mid of the geometry to showcase the temperature distribution and we analyze in the given diagram.

V. CALCULATION:

Waste heat recovery: Waste heat recovery shows the measure of waste heat of the fume's gas consumed by the fuel in the heat exchanger. It is gotten as a level of waste heat recouped which is given by the proportion of heat consumed by the fuel to heat diverted by exhaust gases. This can be determined as follows**.**

Heat carried through the exhaust gas is given by,

$$Q_e = m_e\, c_{pe}(T_2 - T_1)$$

Where,

m_e= Mass of the exhaust gas in kg/sec.

c_{pe} = The specific heat of exhaust gas in J/kg k.

T_2= final temperature of gas in ^{0}C.

T_1= Initial temperature of gas in ^{0}C.

Heat absorbed through the water is given by,

$$Q_w = m_w\, c_{pw}(T_2 - T_1)$$

Where,

m_w= Mass of the water in kg/sec.

c_{pw}= The specific heat of water in J/kg k.

T_2= final temperature of water in ^{0}C.

T_1= Initial temperature of water in ^{0}C.

Then, overall percentage of heat recovery can be calculated as,

$$\%Q_{rec} = (Q_w/Q_e) \times 100$$

Effectiveness of shell and tube heat exchanger: Effectiveness of the shell and tube heat exchanger can be calculated as follows,

$$\varepsilon = \frac{\text{Actual heat transfer}}{\text{Maximum heat transfer}}$$

Actual heat transfer (Q) = $m_h\, c_{ph}(t_{h1}-t_{h2}) = m_c\, c_{pc}(t_{C1}-t_{C2})$

Maximum heat transfer (Q_{max})= $c_h(t_{h1}-t_{C1})$ or $c_c\,(t_{h1}-t_{c1})$

Where, Q_{max} is the minimum of these two values,

So, $Q_{max} = c_{min}\,(t_{h1}-t_{c1})$

LMTD equation: For parallel flow shell and tube heat exchanger LMTD can be calculated as follows,

$$LMTD = \frac{(Q_1 - Q_2)}{\ln(Q_1 - Q_2)}$$

Where, Temperature difference $Q_1 = t_{h1} - t_{c1}$

Temperature difference $Q_2 = t_{h2} - t_{c2}$

VI. RESULTS: After the exporting the model into the CFD-post the analysis has done. In that the results in the condition of contour plot, streamline, vector plot has been defined.

6.1.1 contour plot of the shell and tube heat exchanger:

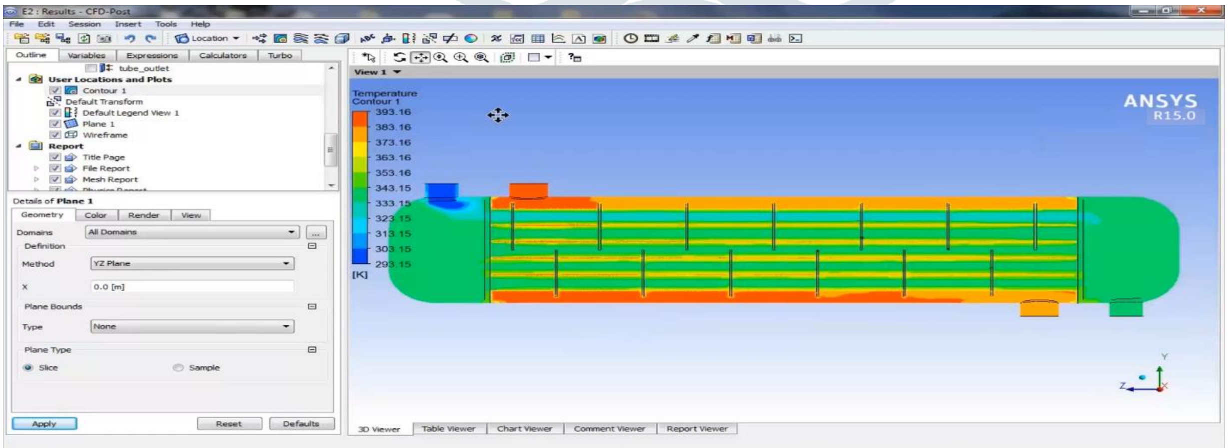


Fig.6.1.1 Temperature distribution for shell and tube heat exchanger in CFD post.

In this contour plot of a CFD-post, the temperature difference has been easily observed by giving the input parameters in the shell and tube heat exchanger. From the above diagram the temperature rise of water can easily be observed and the temperature decreased in the shell side. The value of the outlet parameter is given below, which is found by the doing the analysis part in the Ansys Fluent package 15.0

Table 6.1: Obtained temperature in analysis

Fluids	Inlet Temperature	Outlet Temperature	Temperature difference
Hot fluid (Exhaust gas)	120^{0}C	90^{0}C	30^{0}C
Cold fluid (water)	20^{0}C	60^{0}C	40^{0}C

6.1.2 sectional view of shell and tube heat exchanger at inlet:

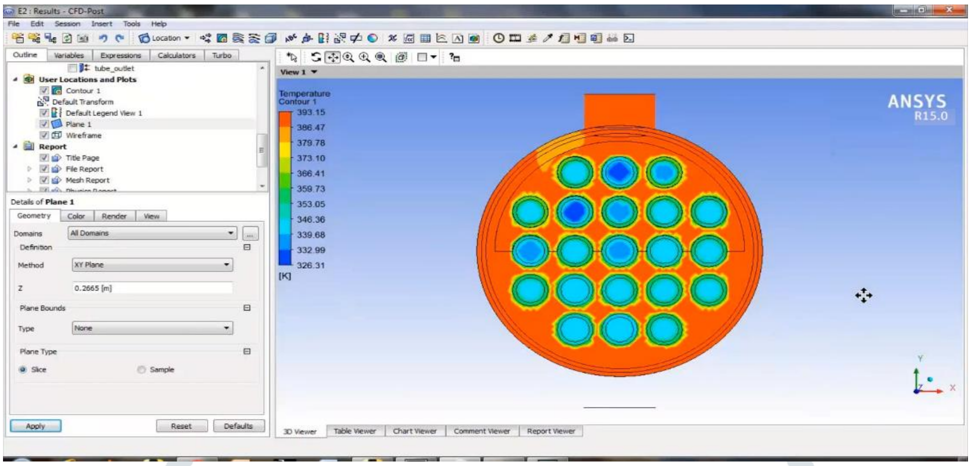


Fig.6.1.2 Inlet temperature of shell(contour1)

6.1.3 sectional view of shell and tube heat exchanger at outlet:

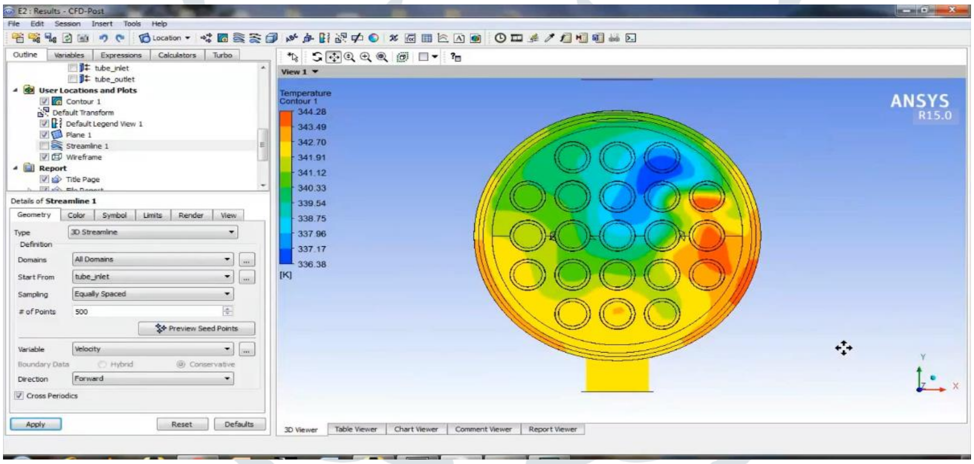


Fig.5.1.3 Outlet temperature of shell(contour1)

6.1.4 Velocity streamline for STHE:

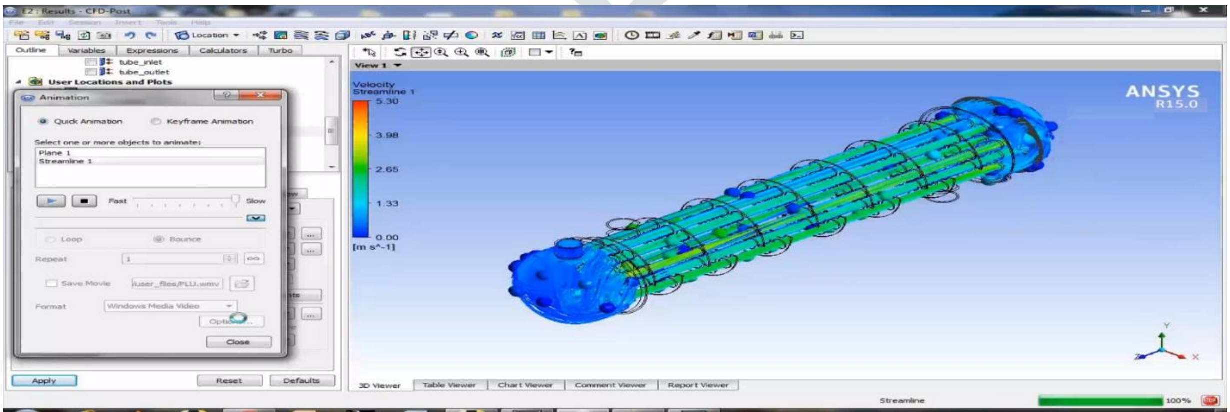


Fig.5.1.4 Velocity Streamline for shell and tube heat exchanger

VII. CONCLUSION: The above experimentation has been done and shows the fluid flow parameters and temperature distribution in the shell and tube heat exchanger. There are some numerical results and data that also has been obtained which are as follows,

The waste heat recovery from the shell and tube heat exchanger is 72.09%

The effectiveness of shell and tube heat exchanger is 0.96.

The LMTD value of shell and tube heat exchanger is 29.41^{0}C.

To show the fluid flow in shell and tube heat exchanger by using computational fluids dynamics in ANSYS Fluent package 15.0

- **Future scope:**

Heat Exchanger have a number of beneficial features. Also, this equipment come in various configuration and type. This makes them ideal for use in a variety of application. Following are some key applications areas, where shell and tube heat exchanger are used:

- HVAC
- Marine Application
- Pulp and Paper
- Refrigeration
- Air processing and Compressor cooling etc.

This paper we finally concluded that the waste heat recovery through this shell and tube heat exchanger due to exhaust gases coming through the diesel engine will reduced in temperature by using this type of heat exchanger. The cold fluids which has been heated through this process can be further utilized in so many industrial purposes. This is also beneficial for the environment because of exhaust gas may harm the environment due to its high temperature.

REFERENCES

[1] Ender Özden and Ilker Tari, (2010). "Shell side CFD Analysis of a Small Shell-and-tube Heat Exchanger", Energy Conversion and Management, Vol. 51, No. 5, pp. 1004-1014.

[2] Rubaiyat, S. N. H., & Bari, S. (2010). Waste heat recovery using shell and tube heat exchanger from the exhaust of an automotive engine (Doctoral dissertation, Bangladesh Society of Mechanical Engineers).

[3] Hossain, S. N., & Bari, S. (2011). Effect of different working fluids on shell and tube heat exchanger to recover heat from exhaust of an automotive diesel engine. In World Renewable Energy Congress-Sweden; 2011; Linköping; Sweden (No. 057, pp. 764-771).

[4] Bari, S., & Hossain, S. N. (2015). Design and optimization of compact heat exchangers to be retrofitted into a vehicle for heat recovery from a diesel engine. Procedia Engineering, 105, 472-479.

[5] Thakur, R. S. D. D. H. (2016). CFD Analysis of Shell and Tube Heat Exchanger. IJSRD-International Journal for Scientific Research & Development, 3(12), 2321-0613.

[6] Patel, D. S., Parmar, R. R., & Prajapati, V. M. (2015). CFD Analysis of Shell and tube heat exchangers-A review. International research journal of engineering and technology, vol102, (09).

[7] Raj, K. T. R., & Ganne, S. (2012). Shell side numerical analysis of a shell and tube heat exchanger considering the effects of baffle inclination angle on fluid flow using CFD. Thermal Science, 16(4), 1165-1174.

[8] Roy, A., & Das, D. H. (2011). CFD analysis of a shell and finned tube heat exchanger for waste heat Recovery applications. International Journal of Mechanical & Industrial Engineering, 1, 77-83.