Proceedings of the
International Conference on Mechanical Engineering and Renewable Energy 2019
(ICMERE2019) 11 – 13 December, 2019, Chittagong, Bangladesh

ICMERE2019-PI-075

THEORETICAL INVESTIGATION OF HEAT TRANSFER ENHANCEMENT OF TRANSFORMER RADIATOR MODIFIED WITH FIN ARRANGEMENT

Mustakim Mamunur Rashid*, Kazi Afzalur Rahman and Farid Ahmad

Department of Mechanical Engineering, Chittagong University of Engineering and Technology
Chattogram-4349, BANGLADESH
*mustakim.mamun.cuet@gmail.com, afzalur99@yahoo.com and faridcuet04@gmail.com,

Abstract-: *Transformer plays a significant role in providing a reliable and useful electricity supply. It is one of the most critical equipment in electric power transmission and distribution systems. Transformer losses are produced by the current passing through resistance on the winding conductors. These losses are converted to heat energy. The majority of high voltage transformers are filled with liquids that work as an electrical insulation as well as a heat transfer medium. The most commonly used liquid in power transformers is mineral oil due to its low cost and good properties. The winding temperature must be kept within a limit so that the temperature dependent properties do not hamper the performance of the transformer. In this study, heat transfer enhancement is investigated for different types of radiator filled with mineral oil. The radiator is modeled in 3D with the help of SOLIDWORKS and thus the oil flow volume is created. The heat transfer calculations and the flow behavior investigations for different types of radiator are done by the ANSYS FLUENT SOFTWARE. The radiator is also modified with assorted fin arrangement to enhance the heat transfer. The results are found in terms of temperature difference, velocity vector, heat flux and streamlines between the inlet and outlet of the radiator. It is found that the bottom oil temperature is suitably dropped for the modified radiator attached with the circular shaped fin and thus, the performance of the transformer is also improved.*

Keywords: Transformer radiator, Fin arrangement, Heat transfer, Radiator oil.

1. INTRODUCTION

A power transformer is the electrical device which is used to change the voltage of AC in power transmission system. The first transformer in the world was invented in 1840s. Modern large and medium power transformers consist of oil tank with oil filling in it, the cooling equipment on the tank wall and the active part inside the tank. Nowadays, transformers play key roles in long distance high-voltage power transmission [1]. Electrical energy is transmitted from power plants to electrical substations. Electricity is more efficiently transmitted at high voltages. Power transformers transmit the electricity at high voltages into substations. Distribution transformers convert the electricity to lower voltages [2]. Wittmaack was stated that chemical degradation in the electrical insulation affects transformer life. Load losses were calculated by using Maxwell electromagnetic field simulation software [3]. Fonte et al. stated that the hot-spot temperature can be reduced by as much 10˚C which has a strong direct impact on the power transformer's lifetime [4]. Joshi and Deshmukh claimed that a new method used in developed new software is more accurate than conventional method to define oil rise and winding rise [5]. Fdhila et al. focused on oil and air flows and the heat transfer in fan-cooled transformer radiators with using ANSYS Fluent 12.0 CFD simulation software. Symmetry boundary condition was used to model of the radiator groups [6]. Gastelurrutia et al. studied on distribution transformers with corrugated walls cooling ONAN (Oil Natural – Air Natural). Fluent V.6.3 software was used implementing the CFD codes [7]. El-Wakil et al. performed a simulation of a step-down transformer including core and windings by using Fluent Software. PISO (the pressure-implicit with splitting of operators) method was used to solve equations [8]. Fernández et al. compared main properties of alternative fluids for power transformers. Mineral oils are widely used in oil-immersed transformers although they are not environmentally friendly and they contain high risk of fire [9]. Transformer losses are produced by the current passing through resistance on the winding conductors. These losses are converted to heat energy. The heat energy produces 80^0-95^0 Temperature. To reduce this temperature of the windings, there used mineral oil as an electrical insulation as well as a heat transfer medium. The hot oil flow through the radiator and the radiator reject the heat by some conduction process and largely convection process. Transformers are cooled by using different cooling modes according to type and power of transformer. Here the transformer cooling system was ONAN (Oil natural Air natural).

This thesis about on simulation of radiator. The main objective of the simulation is to find out the temperature difference of different types of transformer radiator between inlet and outlet by using ANSYS FLUENT (CFD) andto find out the difference of velocity vector, streamline and heat flux between inlet and outlet section of radiators.

2. GOVERNING EQUATION

Computational fluid dynamics (CFD) calculations are based on three main equations; continuity, momentum and energy equations. The form of the continuity equation for laminar flow is shown in Equation (1). Steady-state flow regime was studied in this study. Fluid properties of the oil circulating through the radiator oil pipes do not change over time at any point. Partial derivatives of all quantities with respect to time are zero in steady-state flow regime [10].

$$\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots(1)$$

x, y, z-momentum equations are seen below in Equation (2) - (4), respectively.

x-momentum

$$\frac{\partial}{\partial x}(\rho uu)+\frac{\partial}{\partial y}(\rho vu)+\frac{\partial}{\partial z}(\rho wu) = -\frac{\partial P}{\partial x}+\mu[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}] \ldots\ldots\ldots\ldots(2)$$

y-momentum

$$\frac{\partial}{\partial x}(\rho uv)+\frac{\partial}{\partial y}(\rho vv)+\frac{\partial}{\partial z}(\rho wv) = -\frac{\partial P}{\partial y}+\mu[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}] \ldots\ldots\ldots\ldots(3)$$

z-momentum

$$\frac{\partial}{\partial x}(\rho uw)+\frac{\partial}{\partial y}(\rho vw)+\frac{\partial}{\partial z}(\rho ww) = -\frac{\partial P}{\partial x}-\rho g+\mu[\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}] \ldots\ldots\ldots\ldots(4)$$

The energy equation is shown in equation 5.

$$\rho c_p[(\vec{V}\cdot\nabla)T] = k\nabla^2 T + (\bar{\bar{\tau}}\cdot\nabla)\vec{V} \ldots\ldots\ldots\ldots(5)$$

3. METHODOLOGY

ANSYS Fluent is CFD simulation software to model fluid flow and heat transfer. This study illustrates the fluid flow and heat transfer problem of a transformer radiator. CFD simulation consists of four main parts as modelling, meshing, solution and results. ANSYS Fluent CFD solver uses finite volume method. Modelling and meshing works are specified as pre-processing and solution and results are defined as post-processing.

3.1 Flow Chart of Simulation

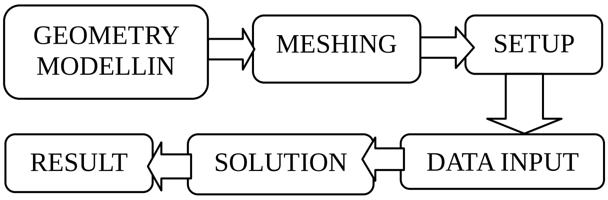

Fig. 1: The flow chart of the simulation system.

3.2 Radiator Geometry

Radiator geometry was modelled in SOLIDWORKS software, and then imported into ANSYS Design Modeler module to obtain a flow volume. Here six models were created for simulations by different oval pipes number and fin arrangement. They are named as model 01 to model 06 according to as shown in Figure 2 to Figure 7.

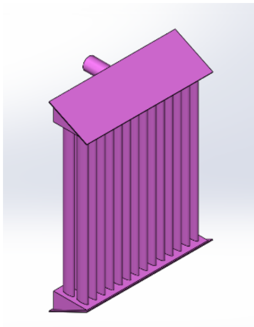

Fig. 2: Radiator with 28 Oval Pipes.

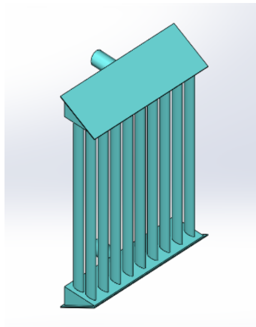

Fig. 3: Radiator with 18 Oval Pipes.

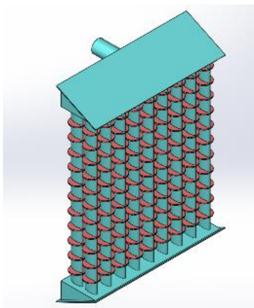

Fig. 4: Radiator with 18 Oval Pipes and Fin Arrangement.

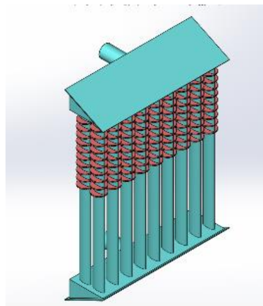

Fig. 5: Radiator with 18 Oval Pipes and upper side Fin Arrangement.

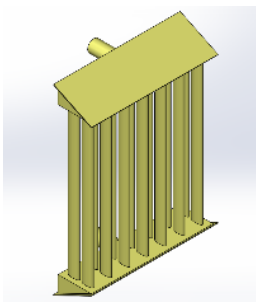

Fig. 6: Radiator with 14 Oval Pipes.

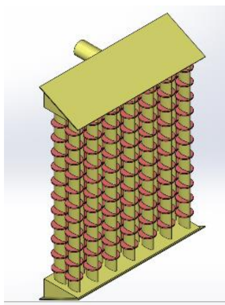

Fig. 7: Radiator with 14 Oval Pipes and Fin Arrangement.

4. NUMERICAL ANALYSIS

ANSYS Fluent is CFD simulation software to model fluid flow and heat transfer. This study illustrates the heat transfer problem of transformer radiator modified with fin arrangement.

4.1 Modelling

Design Modeler is parametric geometry software that provides modelling features for ANSYS analysis. Transformer radiator geometry was modelled in SOLIDWORKS software. The modelled radiator oval pipe has 590 mm length, the oil box length, wide and height are 487 mm, 134 mm and 65 mm respectively and the connecting pipe length 100 mm and diameter 50 mm. Geometry was imported into Design Modeler.

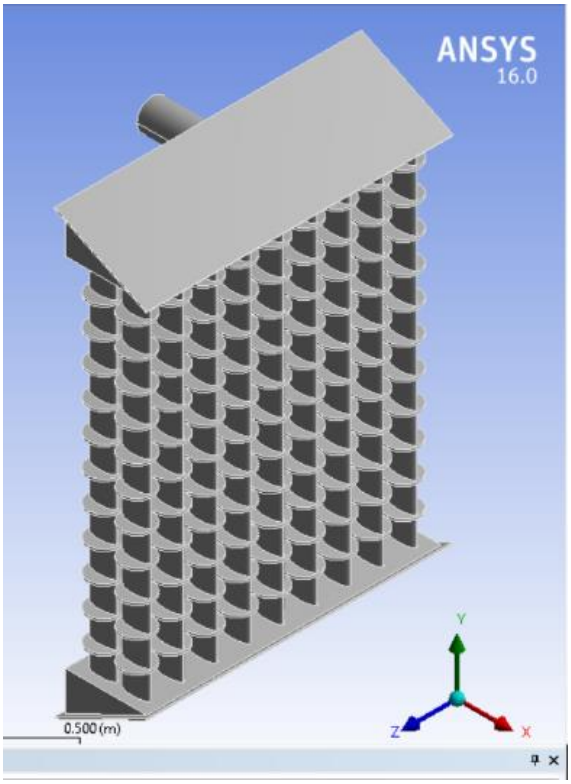

Fig. 8: Imported Radiator Geometry in Design Modeler.

4.2 Meshing

Mesh (grid) module is the most importantpart of CFD simulations.In order to calculate the fluid flow and heat transfer of all body, domains are splitted into small parts.ANSYS Fluent solver uses finite volume method to govern fluid flow and heat transfer equations. Boundary conditions should be defined before meshing operations.

In this study, inlet and outlet boundaries were defined as inlet-velocity and outlet-pressure, respectively. Here, at first select the name of different parts of radiator like as (a)fin (b)oval pipe, (c)hot oil input, (d)output (e)connecting pipe, (f)oil box and that is shown in Figure 9. After selecting name of different parts then it generates the meshing that is shown in Figure 10.

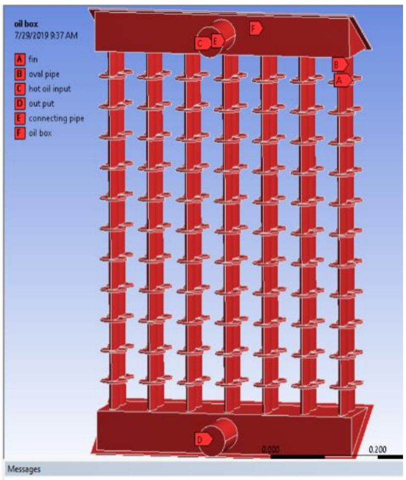

Fig. 9: Name Selection View.

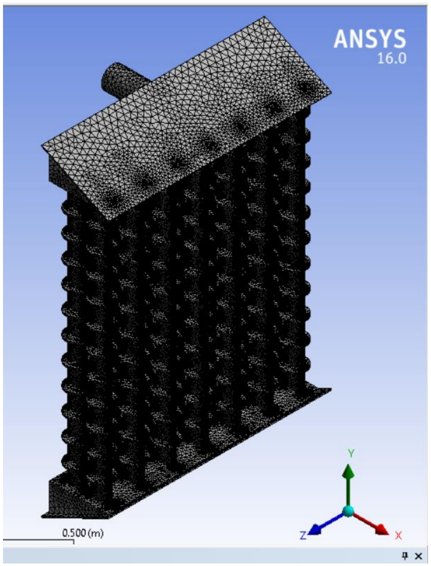

Fig. 10: Mesh of the Radiator Model.

The table 1 shows the number of nodes and number of elements of different models.

Table 1: Number of Nodes and Element of Different Models.

NO.	Nodes number	Element number
Model 01	1256974	6168206
Model 02	816991	4011092
Model 03	959431	4661848
Model 04	976855	4742196
Model 05	638282	3132340
Model 06	763465	3704837

4.3 General Setting

Model settings, material properties, cell zoneconditions, boundary conditions and reference values are the setup procedures. Solution methods, solution controls, monitors, initialization and calculations are performed in solution.

Pressure-based solver was used which is applicable for incompressible flow. Inlet and outlet area are equal and inlet velocity was defined constant. Therefore, mass flow rate at inlet and outlet are equal in each case. The velocity of transformer oil is independent of time at any point. Thus, steady-state flow regime was assumed in this study. Acceleration due to gravity was defined 9.81 m/s^2 along y axis.

4.4 Material Properties

The property of napthanic oil (All property taken in 364 K)	
Name of the Oil:	Napthanic Oil
Chemical Formula:	C_7H_{14}
Density:	827.18 kg/m^3
Specific Heat:	2476.2 J/kg-K
Thermal Conductivity:	0.1249 w/m-K
Dynamic Viscosity:	0.05 kg/m-s
The property of steel	
Name of the Steel:	Mild Steel
Density:	7850 kg/m^3
Specific Heat:	490 J/kg-K
Thermal conductivity:	46 w/m-K

4.5 Boundary Conditions

Velocity of Input Oil:	0.001 m/s
Inlet Temperature of Oil:	364 K
Heat transfer coefficient assumed as in this study	6 W/m^2K
Free stream temperature:	305 K
Heat Generation Rate:	0 w/m^3
Connecting Pipe Wall thickness:	3 mm
Oil Box Wall thickness:	2.5 mm
Oval Pipe Wall thickness:	1.5 mm
Fin Wall thickness:	1.75 mm
Ambient temperature:	305 K

4.6 Solution

Steady-state problem was investigated in this study, SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm was selected in solution methods. Velocity and pressure gradients were calculated to solve momentum equation.Firstly, only the flow was investigated by closing the energy equations under Solution Controls menu to simplify iterations. Before starting calculations, solution is initialized using hybrid initialization. Iterations continue and the monitors were followed until the solution converges. As seen in Figure 11, solution was converged after nearly 498 iterations.

5. RESULTS AND DISCUSSION

ANSYS Fluent CFD simulations use finite volume method to solve continuity, momentum and energy equations. CFD flow volume was created to simulate transformer oil flow in radiator. Fluid flows are classified into two groups as compressible and incompressible flows. Incompressibility is a fluid dynamics property that describes the flow. In this study, incompressible flow was studied. Incompressible flows have small changes in density. Liquids are generally classified as incompressible fluids. Velocity inlet is a boundary condition in ANSYS Fluent CFD software. Velocity inlet boundary condition was used at inlet with constant velocity. Different type of radiators and fin arrangement were applied with constant velocity. Also, inlet temperature was specified as 364 K for all simulations. Convection heat transfer thermal condition was defined at wall. The most important limitation of the study is the mesh procedure. Here it was preferred the SOLIDWORKS software to design geometry instead of ANSYS Design Modeler module. Then, geometry should be updated in Design Modeler and also in Mesh module to avoid creating sharp edges. High performance computers are required to perform the calculations in 3D CFD simulations.

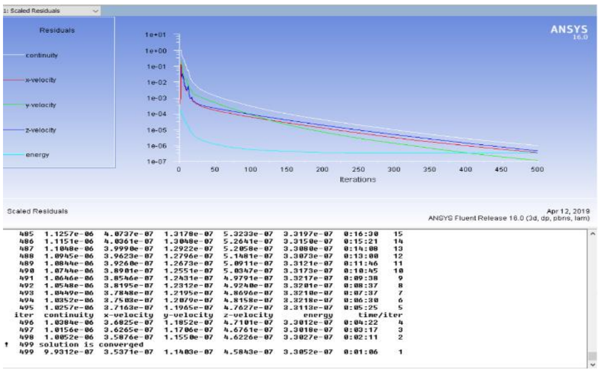

Fig. 11: Radiator Scaled Residual.

5.1 CFD Results

Results module is the CFD post-processing module in ANSYS software to visualize the flow and heat transfer in the simulation model. In this section, napthanic transformer oil will be investigated with visualizing temperature distribution, velocity vectors, heat flux and velocity streamlines of transformer radiator. According to the results, geometry of transformer radiators can be improved.

5.2 Temperature Distribution

Inlet temperature was set to 364 K at inlet boundary for all numerical analysis. The detailed temperature distributions around inlet zones, outlet zones and simulated full model are described in below one by one.

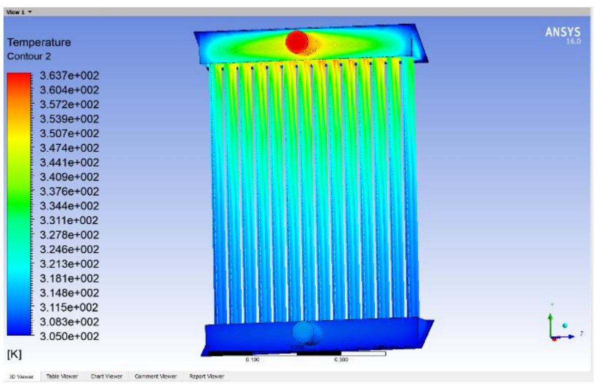

Figure 12: Temperature Distribution of Model 01.

5.2.1 Model No. 01

The Figure 12 is shown that the temperature distribution of model no. 01. Here the figure shows that the napthanic oil temperature is gradually decreased and the temperature reached at free stream temperature. According to legend, red zone is around 364 K and in outlet blue zone is around 313 K.

5.2.2 Model No. 02

The Figure 13 is shown that the temperature distribution of model no. 02. According to legend, red zone is around 364 K andin outlet blue zone is around 320 K.

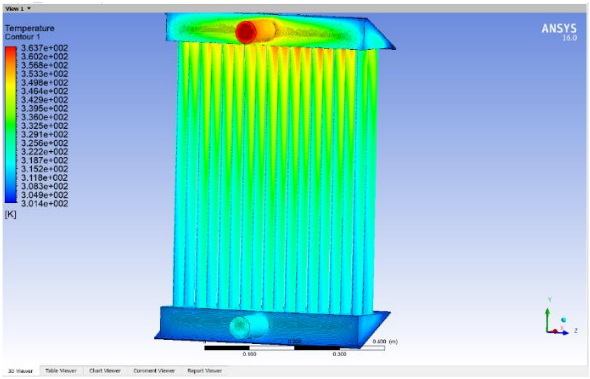

Fig. 13: Temperature Distribution of Model 02.

5.2.3 Model No. 03

The Figure 14 is shown that the temperature distribution of model no. 03. According to legend, red zone is around 364 K and in outlet blue zone is around 318 K. The temperature is decreased 2 K than model no. 02 due to fin arrangement.

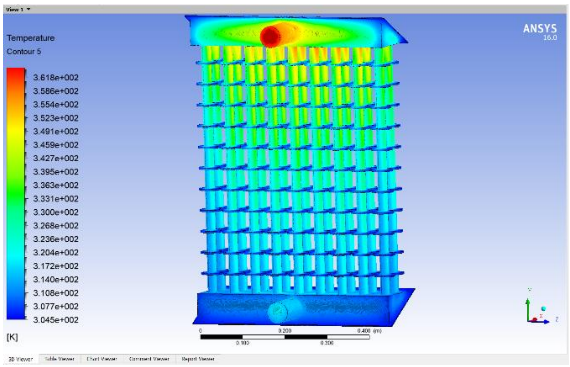

Fig. 14: Temperature Distribution of Model 03.

5.2.4 Model No. 04

The Figure 15 is shown that the temperature distribution of model no. 04. According to legend, red zone is around 364 K and in outlet blue zone is around 317.5 K. The temperature is decreased 2.5 K than model no. 02 due to fin arrangement.

5.2.5 Model No. 05

The Figure 15 is shown that the temperature distribution of model no. 05. According to legend, red zone is around 364 K and in outlet blue zone is around 324 K.

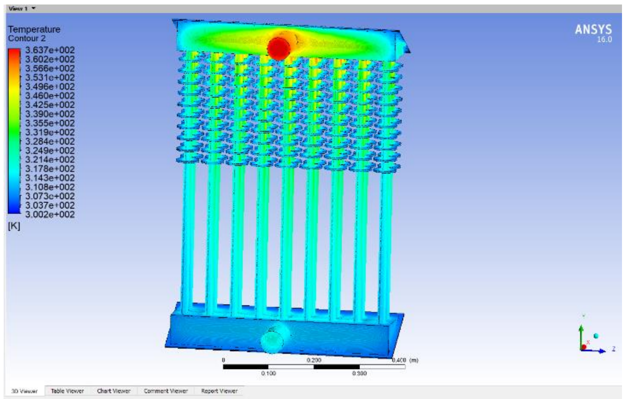

Fig. 15: Temperature Distribution of Model 04.

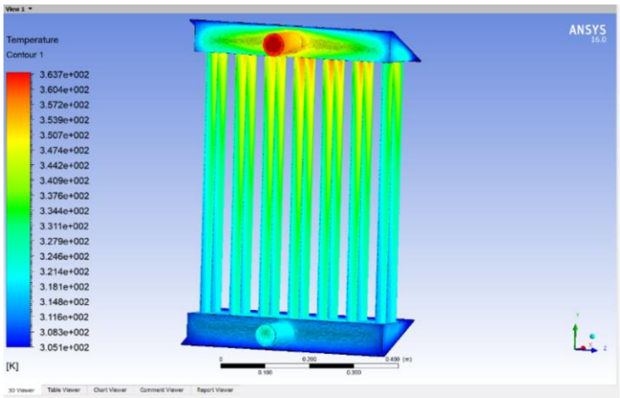

Fig. 16: Temperature Distribution of Model 05.

5.2.6 Model No. 06

The Figure 15 is shown that the temperature distribution of model no. 06. According to legend, red zone is around 364 K and in outlet blue zone is around 322 K. The temperature is decreased 2 K than model no. 05 due to fin arrangement.

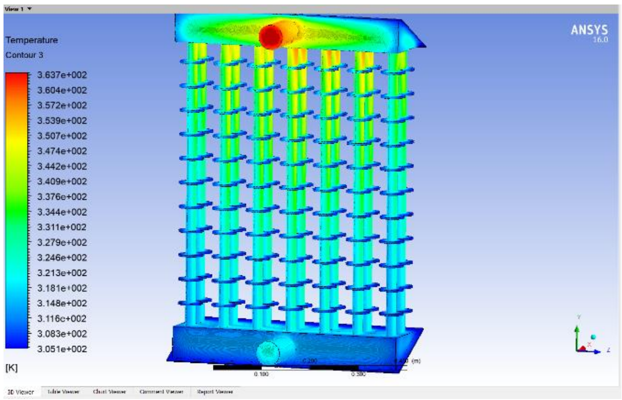

Fig. 17: Temperature Distribution of Model 6.

The table 2 shows the Temperature Distribution of different models.

Table 2: Temperature Distribution of Different Models

No.	Inlet Temperature K	Outlet Temperature K
Model No. 01	364	313
Model No. 02	364	320
Model No. 03	364	318
Model No. 04	364	317.5
Model No. 05	364	324
Model No. 06	364	322

5.3 Velocity Vector

Velocity profile of napthanic transformer oil was investigated by ANSYS Fluent. Velocity vectors were defined as normal to boundary to avoid the reversed flows at outlet boundary. The inlet velocity is same for all models but the outlet center velocity is about double than inlet due to wall velocity is zero and it gradually increase to center. The outlet average velocity is equal to inlet velocity. The Table 3 shows the Velocity Vectors of different models.

5.4 Velocity Streamlines

Streamlines were sketched for this flow to create velocity profile. Although the inlet velocity was 0.001 m/s, it is seen that velocity is decreased to nearly zero inside the oil pipes as shown in Figure 20. This velocity decreasing because of the small thickness that oil ducts have. Especially, at low temperatures napthanic transformer oil cannot flow inside the radiator with this traditional method. Velocity streamlines were shown in Figure 17.

Table 3: Velocity Vectors of Different Models

No.	Inlet Velocity (m/s)	Outlet Velocity (m/s)
Model No. 01	0.001	0.001853
Model No. 02	0.001	0.001854
Model No. 03	0.001	0.001823
Model No. 04	0.001	0.001854
Model No. 05	0.001	0.001856
Model No. 06	0.001	0.001858

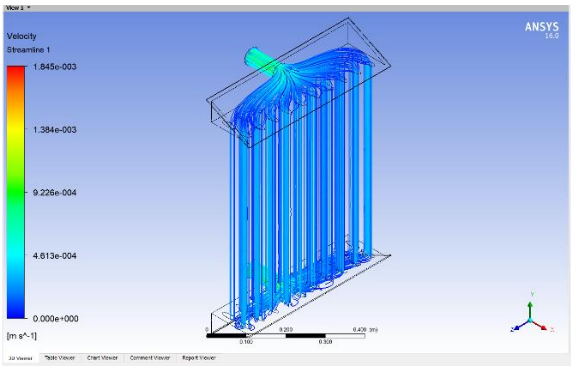

Fig. 17 : Velocity Stream line at full radiator Section.

6. CONCLUSION

The aim of this study is the investigation of transformer radiator temperature of different types of model. Radiator models were simulated with using napthanic transformer oils. From the results of temperature distribution it is found that if the model no 03 is implemented in practical experiment, it will be suitable and cost effective. Furthermore, mineral oils are the most common oil used in oil immersed transformers although they have high risk of fire with 170 ˚C. Also, they are not environmentally friendly. Therefore, non-renewable transformer cooling fluids will be replaced with renewable alternative cooling fluids in the near future.

REFERENCES

[1] Yuhe Jiao, "CFD Study On The Thermal Performance of Transformer Disc Windings Without Oil Guides", KTH School of Industrial Engineering and Management Energy Technology EGI-2012-089MSC EKV915.

[2] P. Picher, F. Torriano, M. Chaabanet al., "Optimization of Transformer Overload Using Advanced Thermal Modelling," in CIGRE Conference, Paris, France A2-305, 2010.

[3] Wittmaack, R. (2014). Thermal design of power transformers via CFD. *11th. World Congress on Computational Mechanics (WCCM XI).*

[4] Fonte, C. M., Lopes, J. C. & Dias, M. M. (2011, 6-9 June). CFD analysis of core type power transformers. *21st International Conference on Electricity Distribution*, Frankfurt.

[5] Joshi, K.U., &Deshmukh, N. K. (2004). Thermal analysis of oil cooled transformer. *Cigre 2004 Session*, Paris, France.

[6] Fdhila, R.B., Kranenborg, J., Laneryd, T., Olsson, C., Samuelsson, B., Gustafsson, A.&Lundin, L. (2011). Thermal modelling of power transformer radiators using a porous medium based CFD approach. *Second International Conference on Computational Methods for Thermal Problems*, China.

[7] Gastelurrutia et al. (2009). Numerical modelling of natural convection of oil inside distribution transformers. *Applied Thermal Engineering, (31)*, 493-505.

[8] El Wakil, N., Chereches, N.C.&Padet,J. (2006). Numerical study of heat transfer and fluid flow in a power transformer. *International Journal of Thermal Sciences*, 45, 615-626.

[9] Fernández, I., Ortiz, A. Delgado, F., Renedo, C.&Pérez,S. (2012). Comparative evaluation of alternative fluids for power transformers. *Electric Power Systems Research, 98*, 58-69.

[10] ÖzbenKAYMAZ et al., "INVESTIGATION OF OIL FLOW AND HEAT TRANSFER IN TRANSFORMER RADIATOR", Izmir Institute of Technology, 2015.