iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 8 **Issue:** IX **Month of publication:** September 2020

DOI: https://doi.org/10.22214/ijraset.2020.31281

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue IX Sep 2020- Available at www.ijraset.com

Prediction of Heat Transfer Coefficient for Condensation Process using CFD by Varying Fluid Velocities

N Suneel Kumar[1], J Durga Venkatesh[2] Dr. B H Varaprasad[3]
[1]PG Student Department of Mechanical Engineering, Aditya Engineering College (A) Surampalem
[2]Assistant Professor, Department of Mechanical Engineering, Aditya Engineering College (A) Surampalem
[3]Head of the Department, Department of Mechanical Engineering, Aditya Engineering College (A) Surampalem

Abstract: Condensation of vapor assumes a critical job in a wide scope of huge scope vitality frameworks. Specifically, steam power plants and HVAC frameworks, which, separately, represent 78% of worldwide electric force age and 10–20% of absolute vitality utilization in created nations, depend on the cycle of fume buildup. Other than steam power plants and HVAC frameworks, proficiency of a few mechanical applications, for example, water desalination, water assortment and warm administration rely upon fume buildup. Along these lines, any improvement in the proficiency of fume buildup cycle can prompt huge vitality reserve funds. The headway of figuring offices has prompted the improvement of cutting edge apparatuses for taking care of different viable building issues. One such progression is the Computational liquid elements (CFD) that utilizes mathematical techniques and calculations to take care of and investigate issues that include liquid streams. In our current work, an endeavor is made to research the impact of stream paces of steam and water in an equal channel condenser on heat move rate utilizing CFD programming instrument is ANSYS Fluent.
Keywords: Condenser, Computational fluid dynamics, ANSYS Fluent.

I. INTRODUCTION

Steam condensers are devices in which the exhaust steam from the steam turbine is condensed by means of cooling medium. Generally cooling water is circulated as a cooling medium which absorbs the heat from exhaust steam. This heat transfer to the cooling water results in condensation. During condensation, the steam changes its phase from vapor to liquid (water) and rejects latent heat. The primary object of a condenser is to maintain a low pressure on the exhaust side of the rotor of steam turbine which enables the steam to expand to a greater extent. This results in an increase in available energy for conversation into mechanical work. The secondary object of condenser is to supply pure and hot feed water to the boiler as the condensed steam which is collected in a hot well can be used over again as feed water for the boiler. The use of a condenser in a power plant is to improve the efficiency of the power plant. This can be achieved by decreasing the exhaust pressure of the steam below atmospheric pressure. Another advantage of using the condenser is that the condensed steam may be recovered to provide a source of pure feed water to the boiler which reduces the water softening capacity to certain extent. The present study focuses on the study of effect of flow rates of steam and water in a parallel pipe condenser on heat transfer rate using CFD software tool is ANSYS Fluent.

II. LITERATURE REVIEW AND OBJECTIVES OF CURRENT STUDY

Jun-De-Li, simulated the condensation of water vapour in the presence of non-condensable gas using computational fluid dynamics (CFD) for the turbulent flows in a vertical cylindrical condenser tube. The CFD results also show that, at least for flows involving high water vapour content, the axial velocity of the gas mixture at the interface between the gas mixture and the condensate film is in general not small and cannot be neglected. Jan havlik etal., describes the influence of flow velocity on the condensation process in a vertical tube. The influence of steam flow velocity on the value of the heat transfer coefficient during the condensation process was evaluated. For the condensation of pure steam, the influence of flow velocity on the value of the heat transfer coefficient begins to be seen at higher speeds, conversely, this effect is negligible at low values of steam velocity. The presence of air significantly reduces the value of the heat transfer coefficient. This drop in the heat transfer coefficient is significant at low velocities; on the contrary, the decrease is relatively small at high values of the velocity. Deepak Kumar Yadav calculated the heat transfer for three-pipe concentric heat exchanger and it was found to be greater than that for two-pipe heat exchanger. The higher heat transfer is because of the increase in area through which heat is transferred. Arul selvan Annamalai etal., conducted CFD analysis of a air cooled condenser heat pipe. The heat pipe attained near steady-state after 60 minutes of operation in both the cases of 50 W and 100 W heat flux provided in the evaporation section. In the present case with air as the cooling medium in the condenser section, the low surface convective heat transfer coefficient is the influencing resistance which affects the performance of the condensing process and sequentially the evaporation process in the heat pipe.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue IX Sep 2020- Available at www.ijraset.com

For efficient operation of the heat pipe, the condenser surface should be exposed with circulating water with high h value or higher heat transfer area is required with addition of fins in the condenser section. A Harsha Vardhan reddy et al, determined the heat transfer by convection in refrigeration by varying condenser length by CFD and thermal analysis. The assessment is out on an air-cooled tube condenser of a vapor compression cycle for refrigeration system. The materials considered for tubes are Copper and Aluminum alloys. In CFD analysis, the heat transfer confident more at condenser length 505mm. In thermal analysis, heat flux is more for copper material at condenser length 405mm. So we can conclude that better material is copper. S Girish et al., modeled and numerically analyzed Surface Condenser in Dr. NTTPS in SOLIDWORKS Flow Simulation. A comparative study of condenser integrated with flower baffles to that of condenser with segmental baffles is done. The effectiveness of the condenser is increased with increasing baffle angles.

III. METHODOLOGY

1) Step-1: Creation of geometry of the structure
2) Step-2: Mesh Development
3) Step-3: ANSYS Fluent model development and computation
4) Step-4: Analysis of results

IV. MODELING CONSIDERATIONS

A. Experimental Setup

The condenser pipe was taken to be made up of copper, as it has good properties compared to other materials which would enhance better heat transfer. The condenser is made up of two concentric pipes, steam in the outer pipe and the water in the inner pipe.

B. Dimensions

Length of the pipe = 1000 mm
Diameter of inner pipe = 80 mm
Diameter of the outer pipe = 150 mm
Thickness of pipe = 1 mm

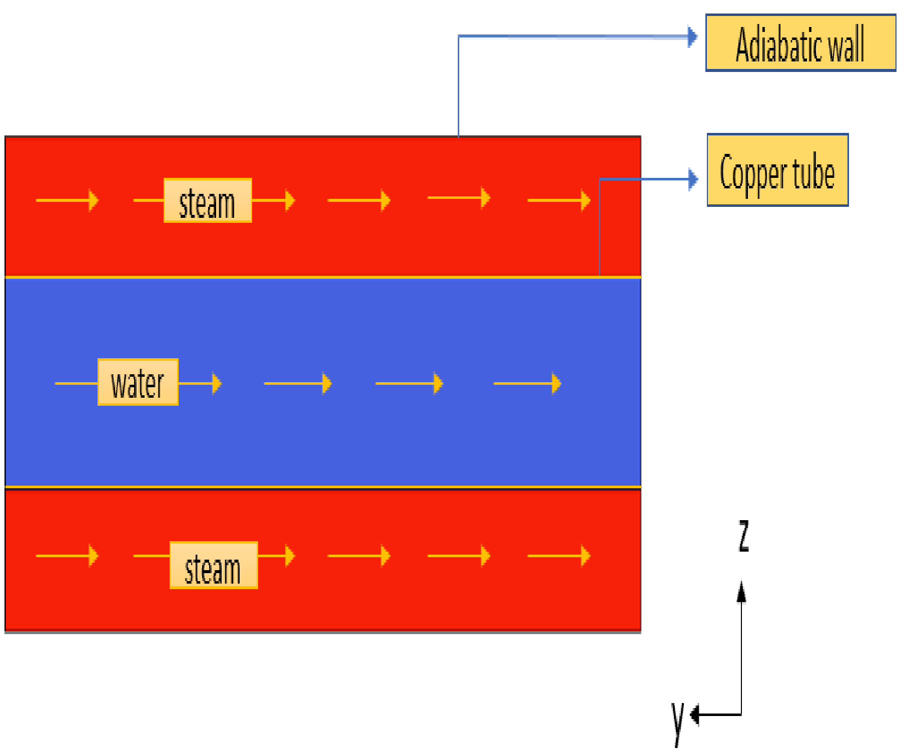


Figure 4.1 Physical Model

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue IX Sep 2020- Available at www.ijraset.com

C. *Properties of Material*

Table 4.1 properties of the materials

Material	copper	Steam	Water
Thermal Conductivity (W/mK)	387.6	0.0261	0.6
Specific heat (J/Kg-k)	381	2014	4182
Density (Kg/m3)	8987	0.5542	998.2
Viscosity (Kg/mS)	-	$1.34e^{-05}$	$1.003e^{-3}$

D. *Mesh Generation*

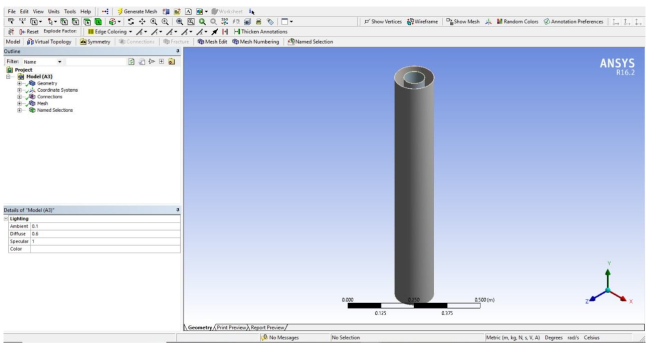


Figure 4.2 ANSYS mesh interface

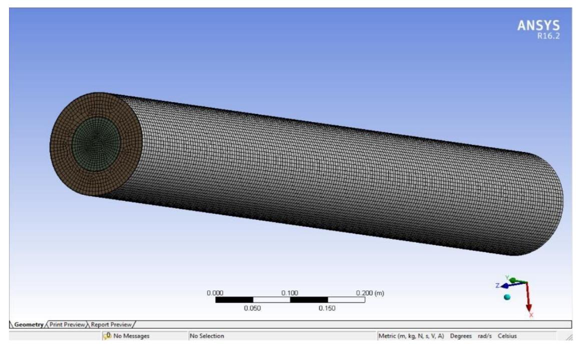


Fig.4.3 Fully developed mesh

E. *Mesh Independent Study*

1) *Step 1:* Run the initial simulation on initial mesh and ensure convergence of residual error to 10-4, monitor points are steady, and imbalances below 1% if not refine the mesh and repeat step 1.
2) *Step 2:* Once we have met the convergence criteria above for our first simulation, refine the mesh globally so that we have finer cells throughout the domain. Generally, we would aim for around 1.5 times the initial mesh size. Run the simulation and ensure that the residual error drops below 10-4, that the monitor points are steady, and that the imbalances are below 1%.
3) *Step 3:* Because the solution is changing with the refinement of mesh, a mesh independent solution has not achieved yet. The mesh has to be refined more, and process should be repeated until to have a solution that is independent of the mesh. And the highest mesh to give this independent solution should be considered in order to decrease the simulation time. The best way to check for a mesh independent solution is to plot a graph of the resultant solution to the mesh size or the number of cells. In our present case, mesh independent study was carried out at different mesh sizes, and the results were illustrated below.

Table 4.2 Mesh independent study

Max face size	Max Size	Nodes	Elements	h (W/m^2K)	% error
0.009	0.02	77288	55373	4952.75	23.85
0.0085	0.02	82110	58442	4886.87	22.2
0.008	0.02	99360	74456	4842.31	21.11
0.007	0.02	115704	91417	4524.84	13.2
0.0065	0.02	176267	141960	4262.41	6.61
0.0055	0.02	250245	208053	4141.59	3.55
0.005	0.02	294069	246408	4139.36	3.53
0.0045	0.02	345689	298456	4137.29	3.52

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue IX Sep 2020- Available at www.ijraset.com

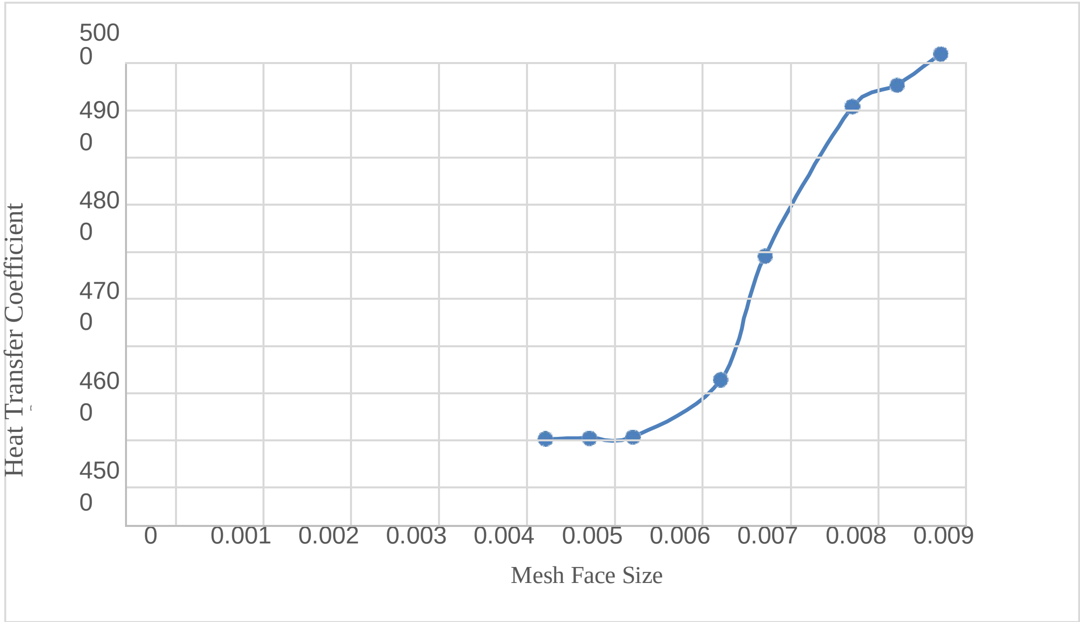


Figure 4.4 Mesh Size Vs Heat Transfer Coefficient Graph

From the graph, the curve gets flattened from a mesh size of 0.0055, as we keep on decreasing the mesh size. This indicates that we have reached a solution value that is independent of the mesh resolution at a mesh size of 0.0055, and for further analysis we can use same cell case, as it will give us a result within the user defined tolerance.

F. Validation

	Overall Heat Transfer Coefficient (w/m^2 k)
Theoretical Result	3998.06
Simulated Result	4141.59
Error in result	3.55%

V. RESULTS AND DISCUSSION

A. Temperature Contour

After the mesh independent study was done, with the determined mesh size, the heat transfer analysis was done on the condenser pipe with same simulation conditions and the results were obtained as follows:

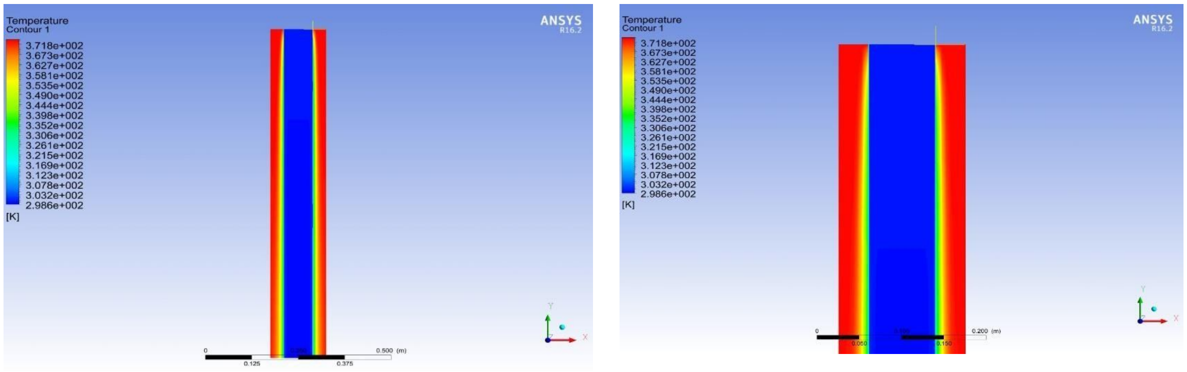


Figure 5.1 Temperature Contours

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue IX Sep 2020- Available at www.ijraset.com

According to Nusselt's analysis of film wise condensation on vertical surface, unless the velocity of the vapour is very high or highly thick liquid film, the motion of the condensate is laminar and the thickness of the condensate film will be a function of rate of condensation of vapour and the rate at which the condensate is removed from the surface of the condenser. It also states that on a vertical surface, the film thickness will increase gradually from top to bottom. The temperature profile obtained shows the formation of film on the surface of the surface of condenser pipe. Hence it satisfies the Nusselt's analysis and shows that the condensation occurred are film wise.

B. Pressure and Velocity Contours

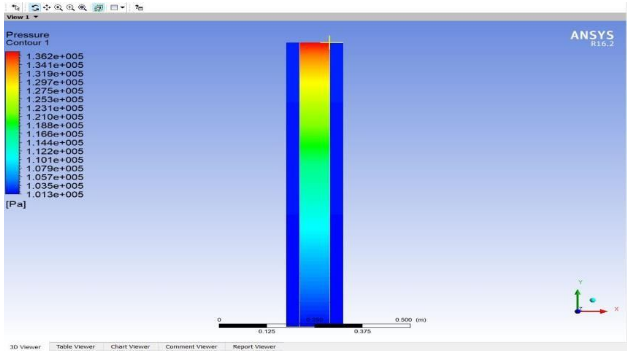


Figure 5.2 Pressure Contour

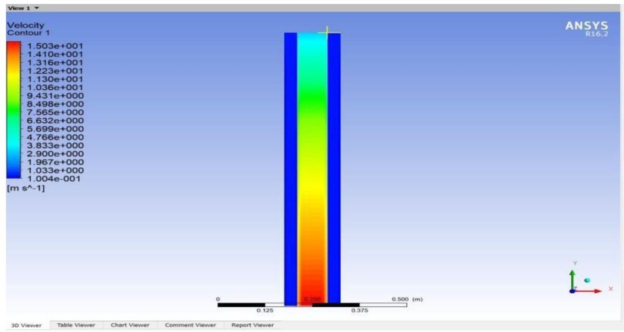


Figure 5.3 Velocity Contour

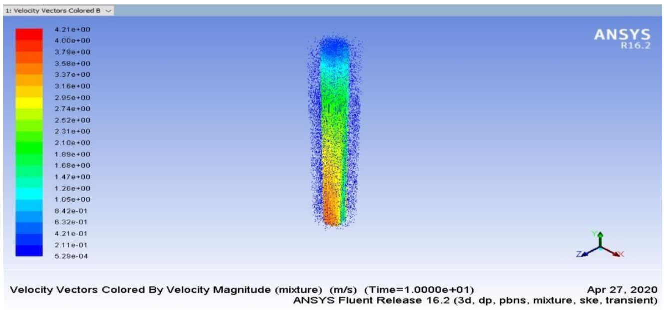


Figure 5.4 Velocity Vector Profile

C. Calculation of Reynolds Number

The Reynolds number is an important dimensionless quantity in fluid mechanics which is used to help predict the flow patterns in different fluid flow situations. At low Reynolds number, flows tend to be dominated by laminar flow, while at high Reynolds number, turbulence results from differences in fluid speed and direction.

	Velocity (m/s)	Reynolds number
Steam	0.1	340
	0.2	680
	0.3	1020
	0.4	1360
	0.5	1700
Water	0.1	8825.04
	0.2	17650.09
	0.3	26475.14
	0.4	35300.19
	0.5	44125.24

Table.5.1. Reynolds number of steam and water at various velocities

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue IX Sep 2020- Available at www.ijraset.com

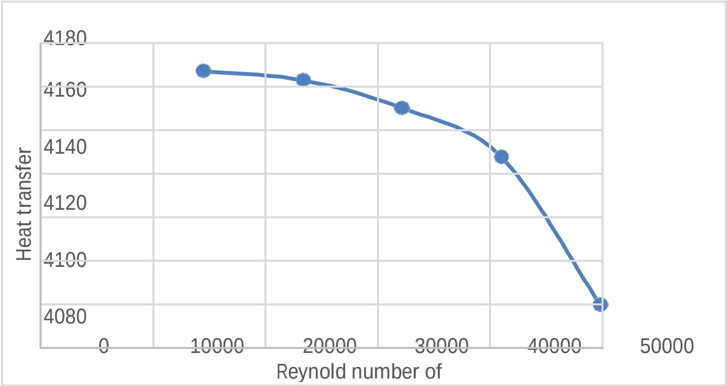


Figure 5.2. Heat Transfer Coefficient Vs Reynolds Number At Varying Water Velocity

The Reynolds number of the steam is found to be varying from 340 to 700 as the velocity increases from 0.1 m/s to 0.5 m/s. this indicates that the steam is following a laminar flow pattern. And for the water, it varied from 8825 to 44125. This indicates the presence of turbulence in the flow of water inside the pipe.

D. Heat Transfer Coefficient

The heat transfer coefficient of the condenser pipe can be calculated form results as follows:

1) Add the locations - point and radial
2) Add expressions for T_m, T_{w1} and h
3) Calculate the heat transfer coefficient (h), at the required location by setting the coordinates in location.

The heat transfer rate is found to be less at lower steam velocities since a higher heat transfer coefficient is obtained at 0.1 m/s steam velocity. The highest heat transfer coeffient is at lowest steam velocity because at low velocities, the turbulence is more, which favors the efficient heat transfer by increasing the friction between the fluid flow and the condenser walls by forcing the steam against the wall of condenser pipe. By varying the water inlet velocities, the results shown very less variation among the results. This shows that the water flow plays less significance in heat transfer rates. However since the graph shows negligible change from 0.3 m/s water inlet velocity, it was considered as economical.

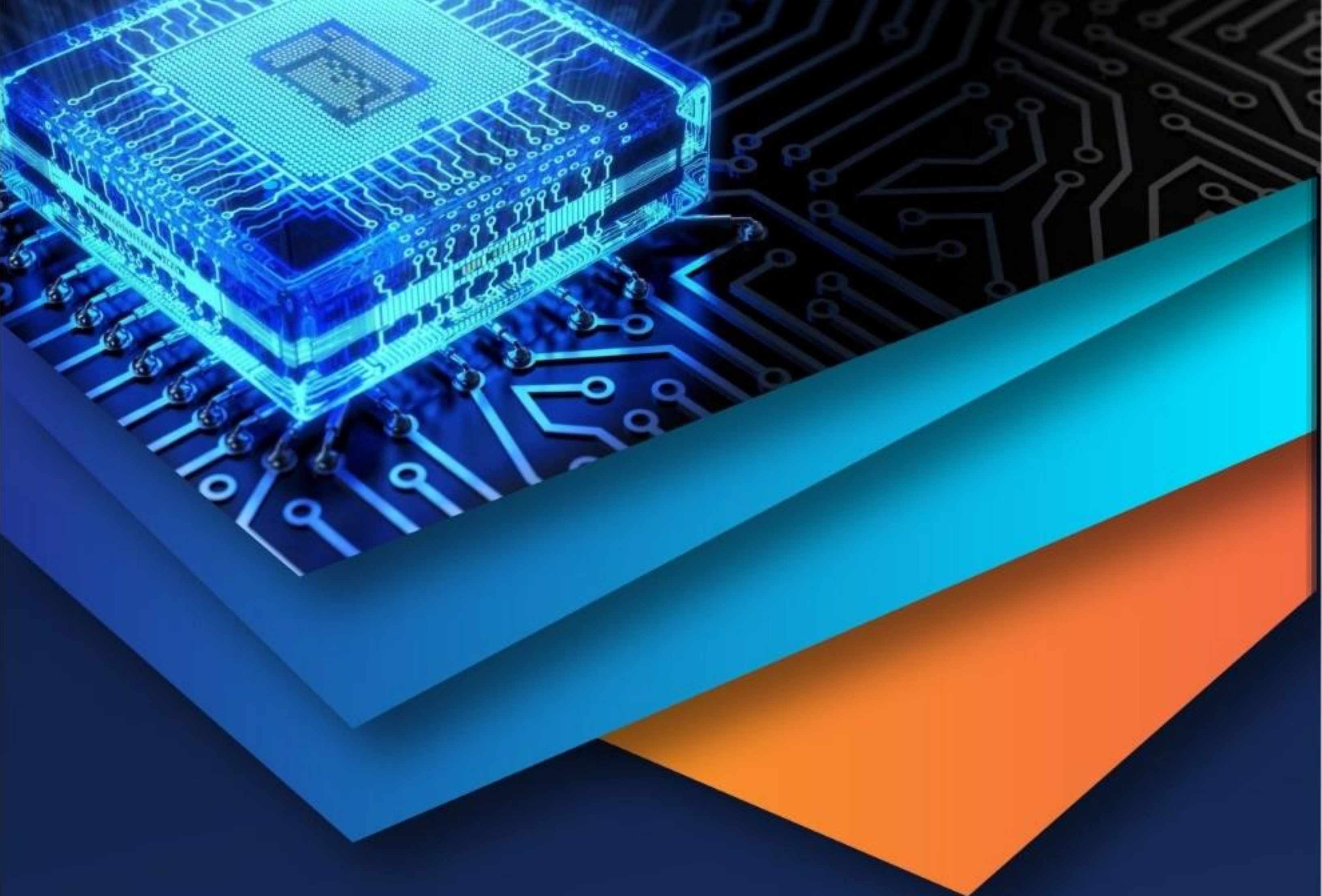
VI. CONCLUSIONS

A. In this study, condenser pipe was designed and analyzed using ANSYS fluent software. All the results are noted down and graphs are drawn for all the values where heat transfer coefficients are taken on the y-axis, velocities on x-axis.
B. The mesh independent study was taken for the problem and the independent mesh was observed at 0.0055 face size with 250245 nodes and 208053 total number of elements. The obtained heat transfer value at this meshing conditions reported a heat transfer value of 4141.59 w/m2k with 3.55% deviation from theoretical value.
C. The heat transfer analysis of the condenser pipe at different steam velocities was done and it was found that the heat tranfer rate is higher at lowest steam velocity. And the inlet velocities of steam and water at 0.1 m/s and 0.3 m/s respectively were found to be economical with 4761.2 w/m2k heat transfer coefficient.

REFERENCES

[1] Varun kalra, “CFD validation and scaling of condensation heat transfer", 2012.
[2] Jun-De Li “CFD simulation of water vapor condensation in the presence of non- condensable gas in vertical cylindrical condensers” , 2012
[3] Jan havlik, Tomas dlouhy “Effect of flow velocity on process of air steam condensation in a vertical tube condenser”, 2017
[4] Deepak Kumar Yadav, “Analysis of concentric pipe heat exchanger”, 2017
[5] Arul selvan Annamalai, velraj ramalingam “Experimental investigation and CFD analysis of a air cooled condenser heat pipe”, 2011
[6] A harsha vardan reddy “Design and analysis of condenser using 3D modelling software”, 2018
[7] S Girish, M surya Prakash, P Geeta Krishna, K Lavanya “Analysis of a condenser in a thermal power plant for possible augmentation in its heat transfer performance”, 2017
[8] Yasuhiro Kudo, Kyosuke Nakaiso, Akio Miyara, "experimental study on boiling and condensation heat transfer in a horizontal mini channel", 2016.
[9] Qingpu Li, Leren Tao, Lei Li, Yongpan Hu, Shengli Wu, "experimental investigation of the condensation Heat transfer coefficient of R134a inside horizontal smooth and micro- fin tubes", 2017.

[10] Wen Wang, Xun Wang, "experiments of condensation heat transfer in micro channel heat exchanger", 2010.
[11] Yanfei Liu, Xiaotian Han, Chaoqun Shen, Feng Yao, Mengchen Zhang, "experimental study on the evaporation and condensation heat transfer characteristics of a vapor chamber", 2018.
[12] Jan Havlik, Tomas Dlouhy, "effect of flow velocity on the process of air-steam condensation in a vertical tube condenser", 2018.
[13] Xuwen Wang, Qiusheng Liu, Zhiqiang Zhu, Xue Chen, "experiments of transient condensation heat transfer on the heat flux sensor", 2015.
[14] Shah Ketul D, H.A.Vaidhya, T.B.Patel, "CFD analysis of condensing heat transfer through mini channel", 2015.
[15] N. Padoin, C. Soares, "CFD modelling of steam condensation in industrial pipes", 2014.
[16] Ladislav Vyskocil, Josef Schmid, Jiri Macek, "CFD simulation of air-steam flow with condensation", 2014.
[17] Nidhi R. Singh, Rashed Ali, "CFD analysis of condensation heat transfer in helical coil heat exchanger", 2018.
[18] Lalatendu Pattanayaka, Biranchi Narayana Padhib, Bibhakar Kodamasingh, "thermal performance assessment of steam surface condenser", 2019.
[19] H.W.Hu, G.H.Tang, "theoretical investigation of stable dropwise condensation heat transfer on a horizontal tube", 2002.
[20] Manjinder Singh, Nilesh D.pawar, "Modelling and simulation of dropwise condensation", 2019.
[21] Mohammad saraireh, "Heat transfer and condensation of water vapor from humid air in compact heat exchangers", 1986.
[22] RC SACHDEVA, “fundmentals of engineering heat and mass transfer”, 2017
[23] ANSYS fluent theory guide, ANSYS.inc, 2013.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY