HBRP PUBLICATION

Advancement in Mechanical Engineering and Technology
Volume 5 Issue 1

Flow over Condenser Pipes with Fins: CFD Analysis

***P.Anusha*[1*]*, M.Naga Swapna Sri*[2]**

[1,2]*Assistant Professor*

Department of Mechanical Engineering, P V P Siddhartha Institute of Technology, Vijayawada, India.

***Corresponding Author**

E-mail Id:-*anusha.peyyala@pvpsiddhartha.ac.in*

ABSTRACT

As a part of this project, CFD analysis is performed on pipe with fins for various header sections. Heat exchangers are used in a number of sectors, and one such heat exchanger (a pipe with fins) is considered in this paper. It is created to meet the industry's business requirements. The condenser is designed using the CREO parameters programme. The design procedure resulted in a condenser with 22 tubes, 20mm outer diameter, 18mm inner diameter, and a length of 90mm.Because the design technique does not specify the type of to be utilized header, investigated 3 different headers that provide equal velocity in the tube's entrance. Various geometries were used in multiple points of header's input nozzle Optimal location of the, which might be used for equal liquid and velocity distribution across each and every tube, is done using CFD models. The major goal of this project is to employ commercial Computational Fluid Dynamics [CFD] software to verify the planned condenser. CREO parameters 3.0 software is used to create a symmetric representation of the simplified geometry of a condenser [Pipe with Fins] for simulation purposes. The velocity, pressure, and temperature characteristics are calculated using the fin as a rectangular plate and air and water as fluids. The procedure is then repeated for the plate with a hole.

Keywords:*-CFD, Pipe with Fins, Condenser, Velocity, Pressure, Temperature.*

INTRODUCTION

Computational Fluid Dynamics [CFD] is a wonderful concept which uses problematic analysis to solve fluid flow problems. Softwares will perform the necessary calculations to model liquids and gases interactions with surfaces specified by boundary equations. Due to latest advancements, software's are now handling complex applications like steady/unsteady, laminar/turbulent fluid flow applications. In the early days, these software's are used for flight tests and also for multiple applications in various sectors. The best alternatives for planning and determining the efficiency level of HVAC systems are often derived using this simulation. To guarantee that the design criteria are met, data from CFD analyses can be utilized to validate a variety of design elements. CFD modeling also helps with the design verification for variety of systems. The CFD model can also be used to spot weaknesses in HVAC design or product failures.

By modifying the exterior material, Praveen Kumar Kanti et al [01] investigated work related to counter flow heat exchangers without baffle plates. According to Amol S. Niphade et al. [02Heat exchangers are so important in practise, especially those that involve chemical reactions and phase changes, to develop cost effective tools.Ankit Uppal and colleagues [03] determined ideal configuration of baffle shape by seven types of baffle layouts for enhancing HT. This analysis was carried out using ANSYS 14.5 Fluent, a three-dimensional finite volume based CFD tool. A heat exchanger tube composed of Al and Cu with a length of 0.1m and a diameter of 0.01m was considered. All computations were done with a Reynolds number of 10,000. Greatest heat transfer is possible for rectangular and triangular shaped baffle surfaces. The reason for the highest heat transfer rate was that baffles increased turbulence by allowing more mixing of fluid layers, which boosted heat transmission through the heat exchanger tube.

A twin pipe heat exchanger's performance rate is shown. by Sk.M.Z.M.Saqheb et al[04]. By altering the materials that use the heat input from the refinery's waste steam recovery process. CATIA and GAMBIT are used to design double pipe heat exchangers. ANSYS is used to perform CFD analysis. The final results were achieved using three different materials: steel, aluminium, and copper.The project of Kranthi Kumar Mamidala et al [05] intends the % inaccuracy in experimental and analytical data by analysing the heat flow pattern of a heat exchanger using Fluent Software.

In industrial processes, exchangers of heat are used for heat recovery fluids, according to Jibin Johnson et al [6]. Heat exchangers are designed to play a certain purpose in any application. Even though the multiple equations are available, validating the design with these equations takes time. The analytical design of the exchanger was done in this work depending on CFD simulation results. Traditional k-modeling was used in this paper's CFD research.

The work of Swapnaneel Sarma et al [7] was done with the goal of predicting the efficiency of a waste heat recovery heat exchanger with fluent CFD, and the findings were compared to existing experimental values. The performance of the heat exchanger was determined for tri angular and the results were compared with traditional rectangular fins.

CFD ANALYSIS PROCESS

The standard approach for performing a CFD study is outlined here to help you understand the many components of a CFD simulation. The technique includes the following steps:

1. Create a diagram of flow for the problem
2. Geometry and Flow Domain Modeling
3. Set the Boundary and the Starting constraint values
4. Grid generation
5. SimulationStrategy
6. Parameters/ Files for Input
7. Execute the Simulation
8. Monitoring
9. post-process the simulation results.
10. Make observations between the outcomes

CONDENSER DESIGN [Figure 1]

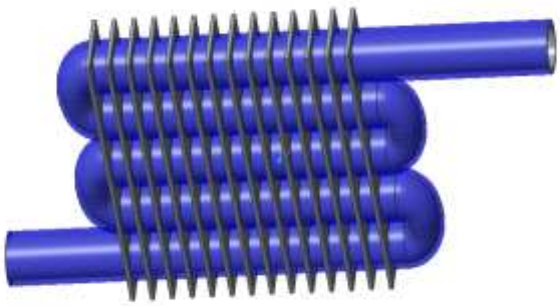


Fig.1:-*Condenser Design*

Parameters

The pipe's outside diameter: 20mm
Pipe inner diameter : 18mm
Length of the fin : 12mm
Width of the fin : 16mm
Thickness : 1mm

RESULTS AND DISCUSSION

The CREO parameter 3.0 programme is used to create a symmetric image of the simplified geometry of a condenser [Pipe with Fins] for simulation purposes.

Initially, by considering fin as a rectangular plate and air as fluid analyzed the variation of parameters either by keeping velocity as constant or temperature as constant.

FLUID AS AIR [Figure 2]

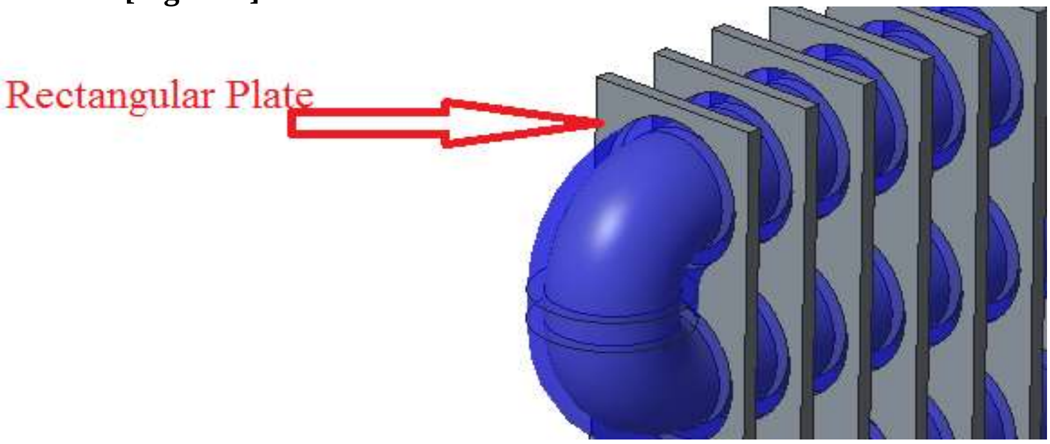
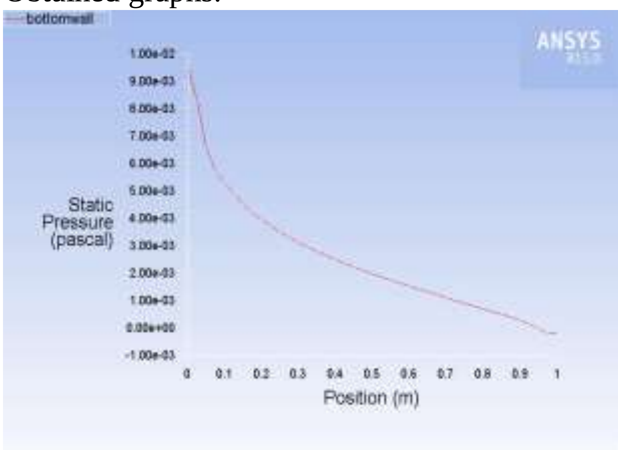


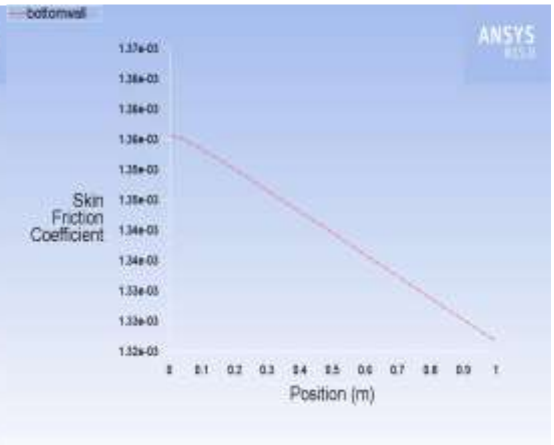
***Fig.2:**-Laminar Flow over a Rectangular Plate considering Fluid as Air*

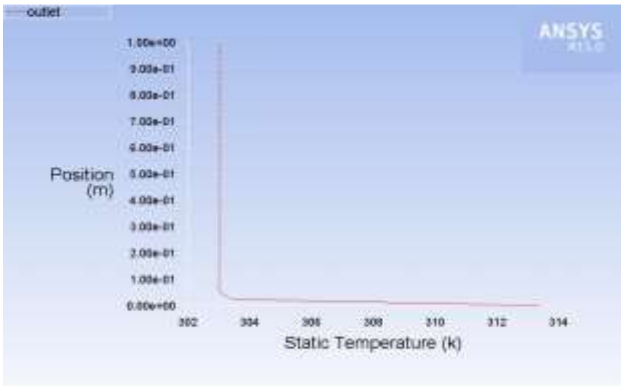
Analysis 1: CFD ANALYSIS OVER A PLATE, Fluid as AIR

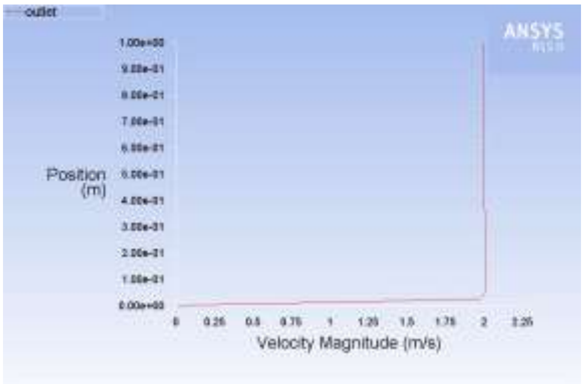
<u>CASE 1: FLUID AS AIR BY KEEPING TEMPERATURE AT 323K & VARYING VELOCITY 2m/s:</u>

Obtained graphs:



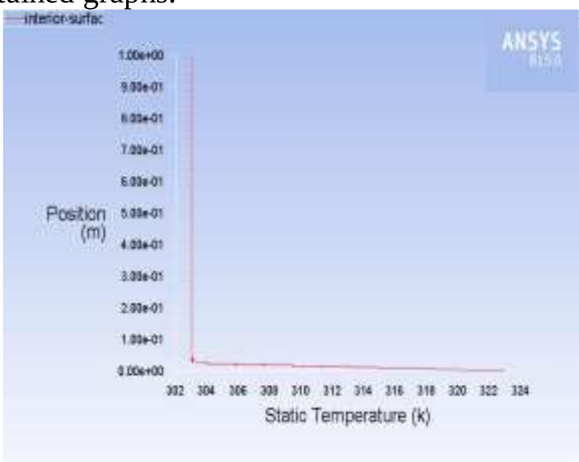


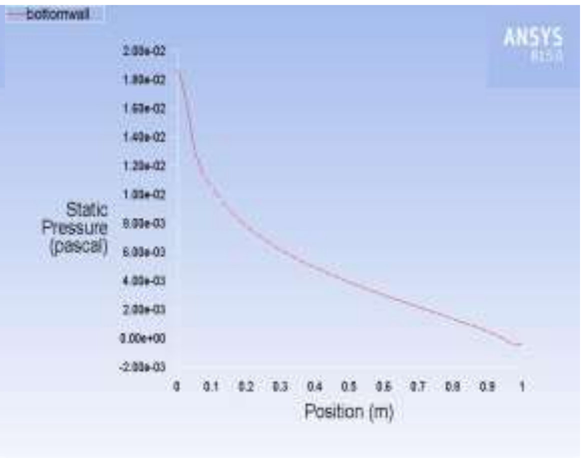


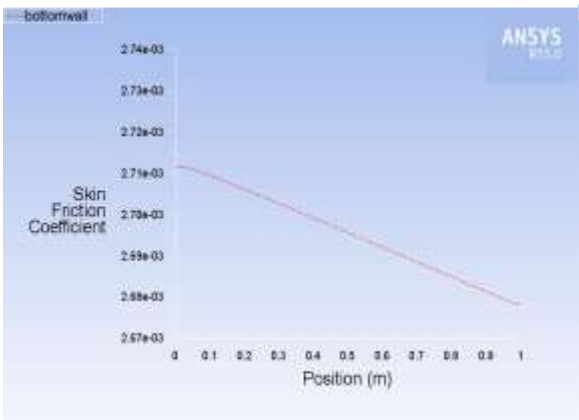


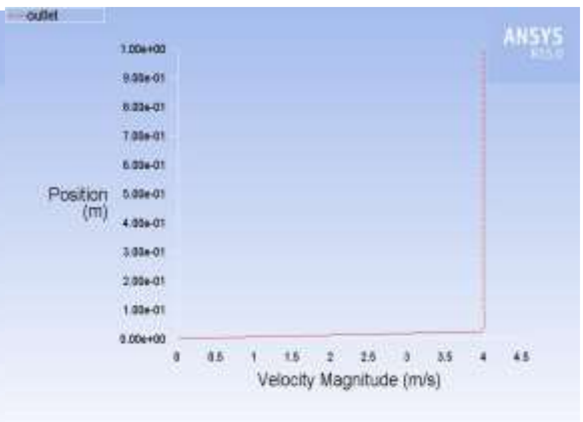
<u>CASE 2: FLUID AS AIR BY KEEPING TEMPERATURE AT 323K & VARYING VELOCITY 4m/s:</u>

Obtained graphs:



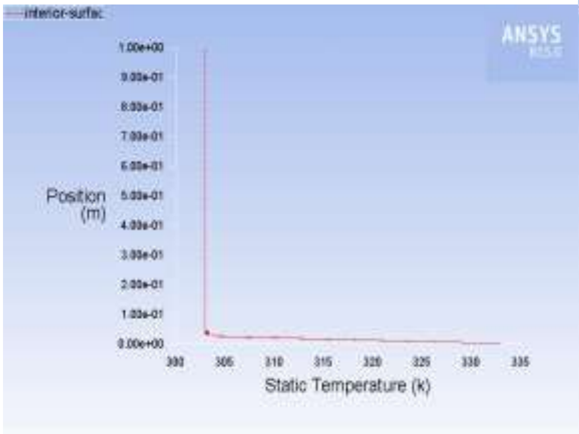


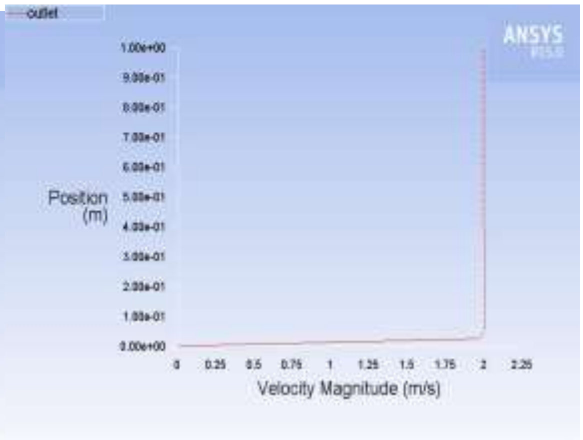


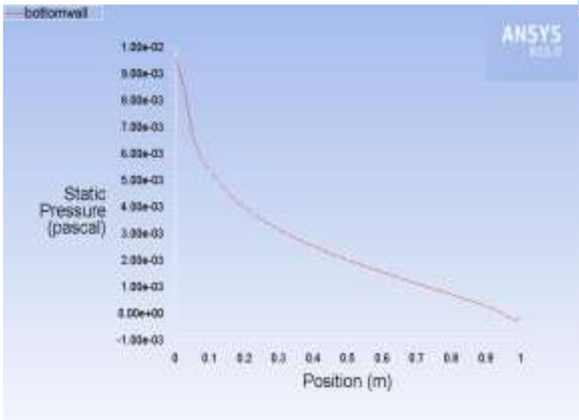


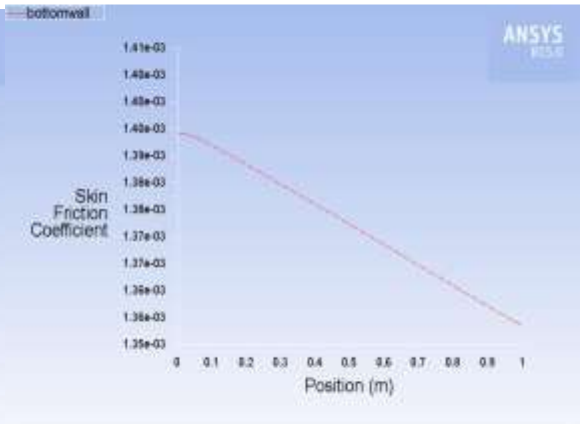
<u>CASE 3: FLUID AS AIR BY KEEPING VELOCITY 2m/s & VARYING TEMPERATURE 333K</u>

Obtained graphs:



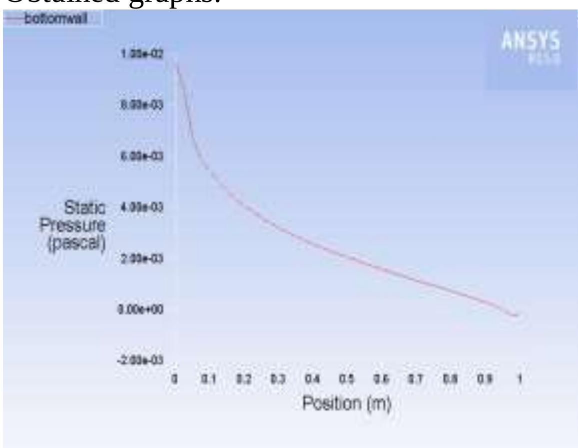


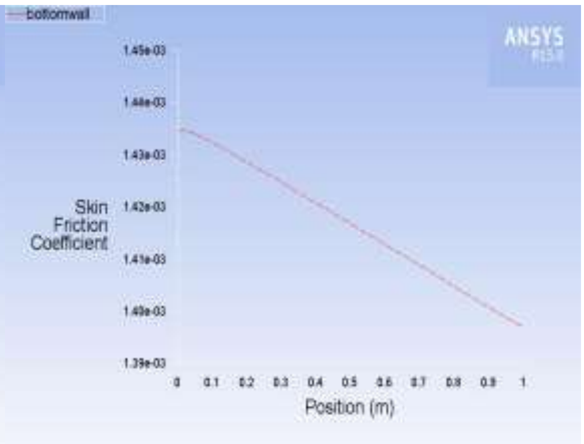


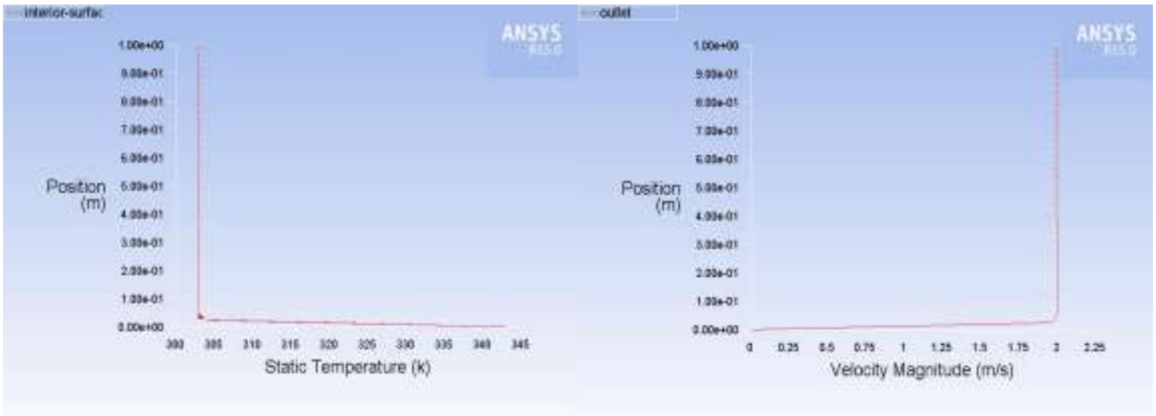


CASE 4: FLUID AS AIR BY KEEPING VELOCITY 2m/s & VARYING TEMPERATURE AT 343K.

Obtained graphs:



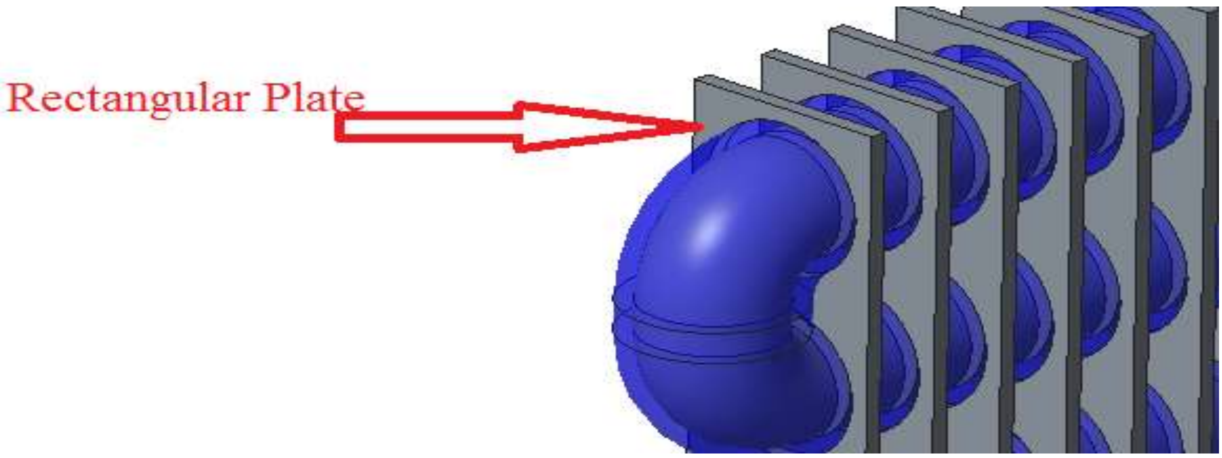




Analysis 2: CFD ANALYSIS OVER A PLATE, Fluid as WATER

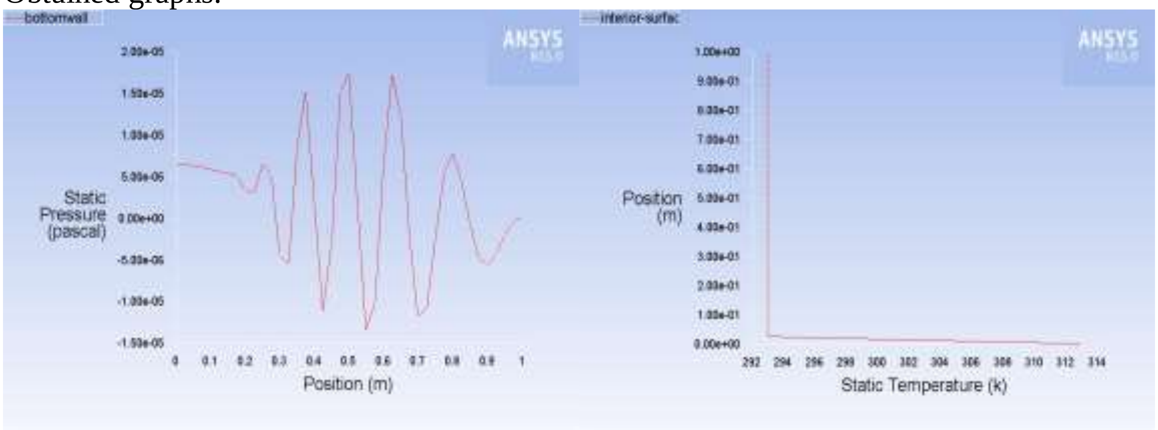
By considering fin as a rectangular plate and water as fluid analyzed the variation of parameters either by keeping velocity as constant or temperature as constant.

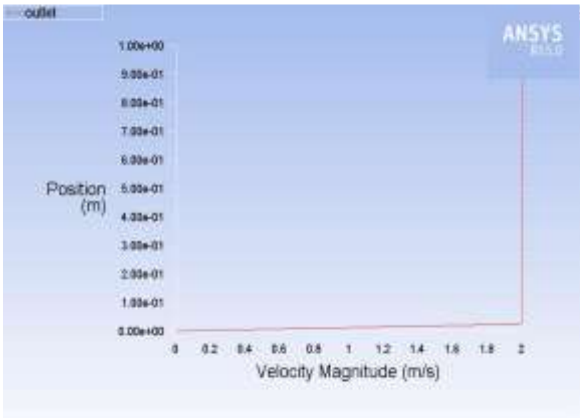
FLUID AS WATER

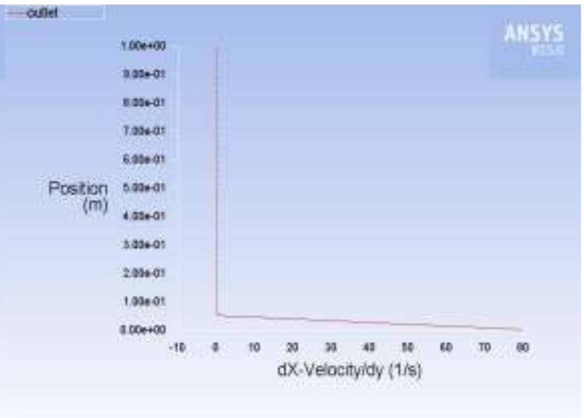


CASE 5: FLUID AS WATER BY KEEPING TEMPERATURE CONSTANT & VARYING VELOCITIES 2m/s

Obtained graphs:

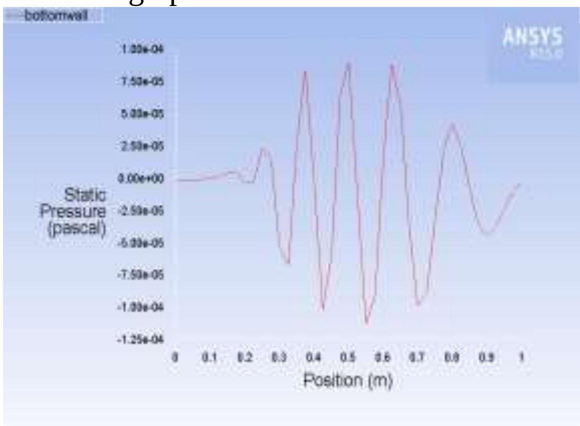


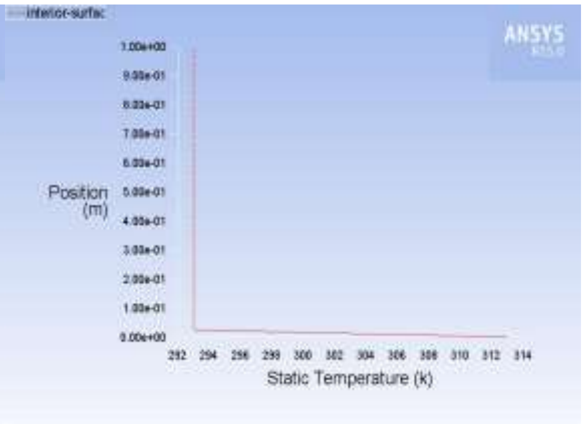


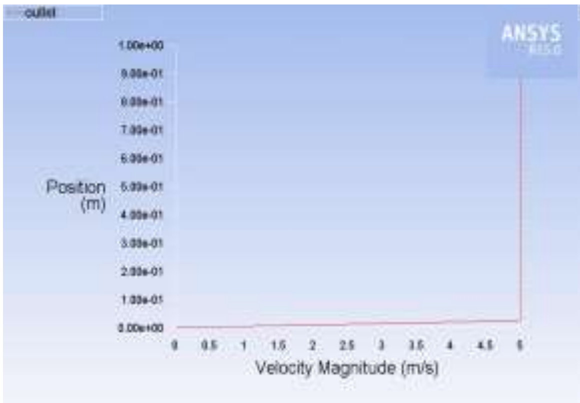


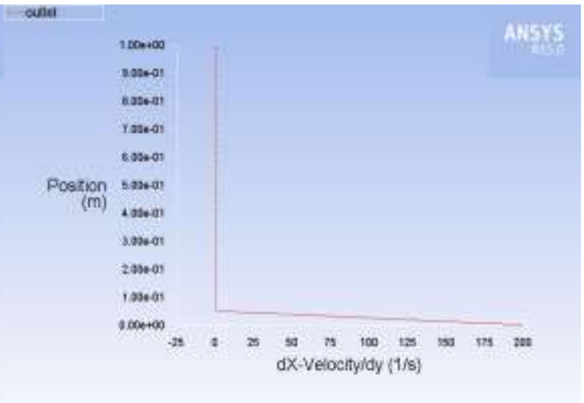
CASE 6 : FLUID AS WATER BY KEEPING TEMPERATURE CONSTANT & VARYING VELOCITIES 4 m/s

Obtained graphs:



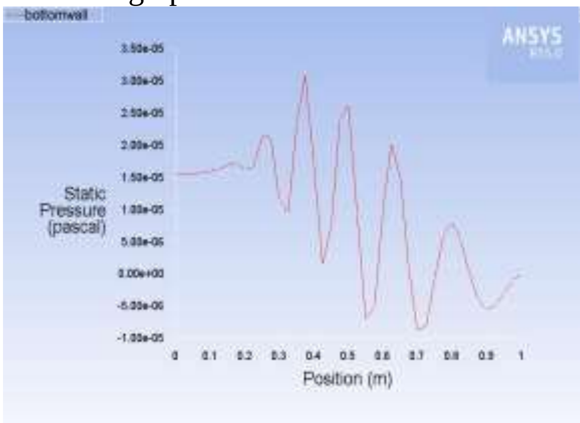


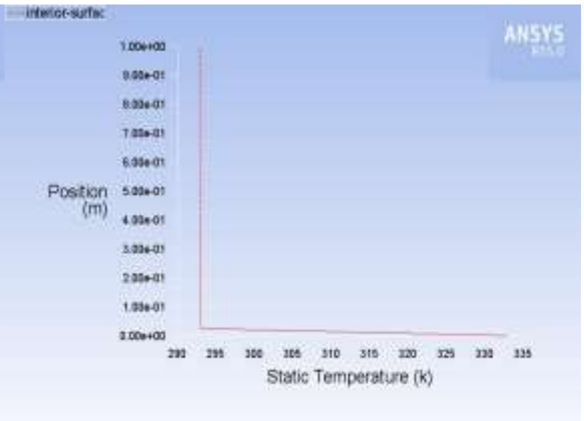


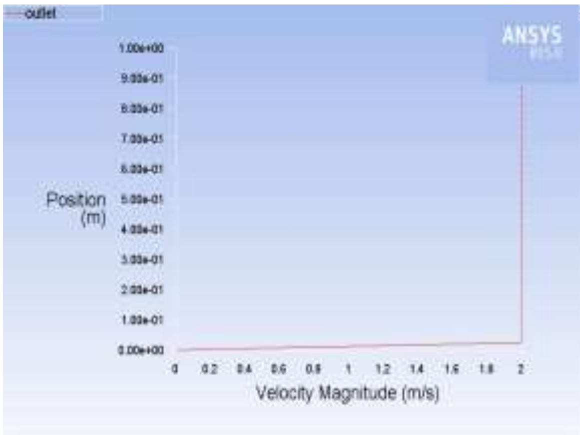


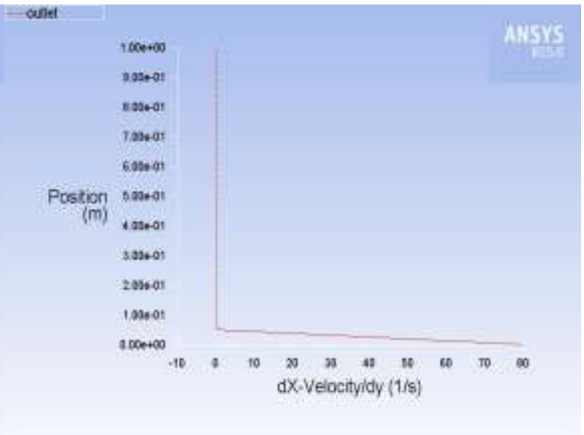
CASE 7 : FLUID AS WATER BY KEEPING VELOCITY CONSTANT & VARYING TEMPARATURES 323k :

Obtained graphs:



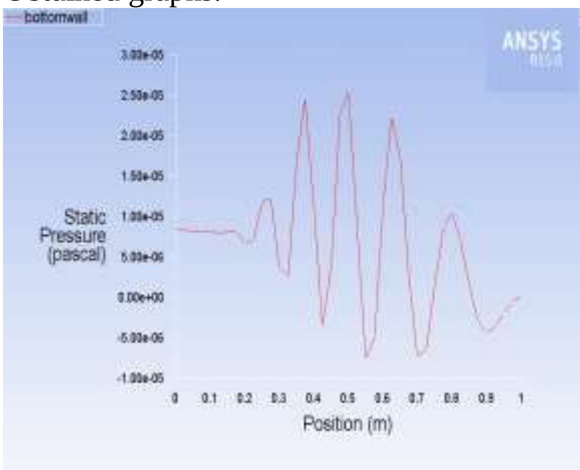


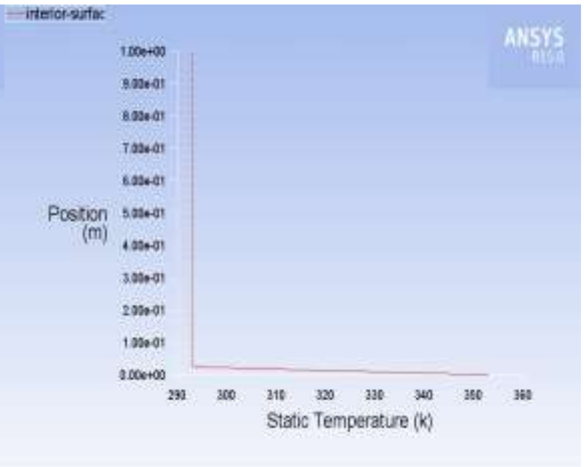


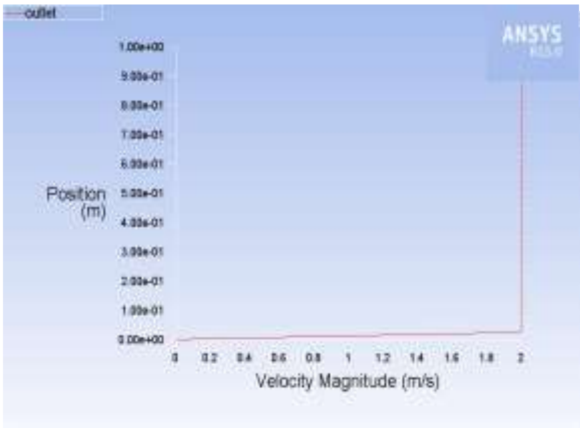


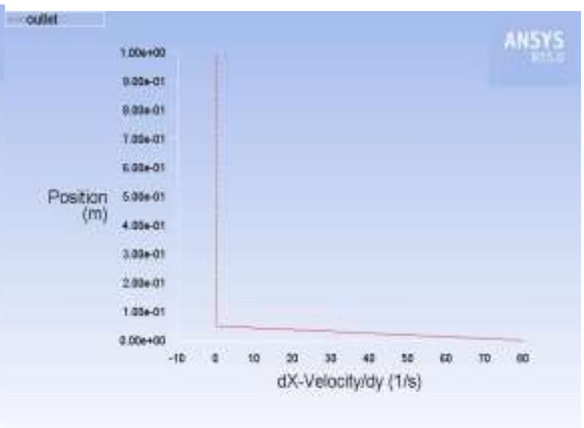
CASE 8: FLUID AS WATER BY KEEPING VELOCITY CONSTANT & VARYING TEMPARATURES 343k :

Obtained graphs:





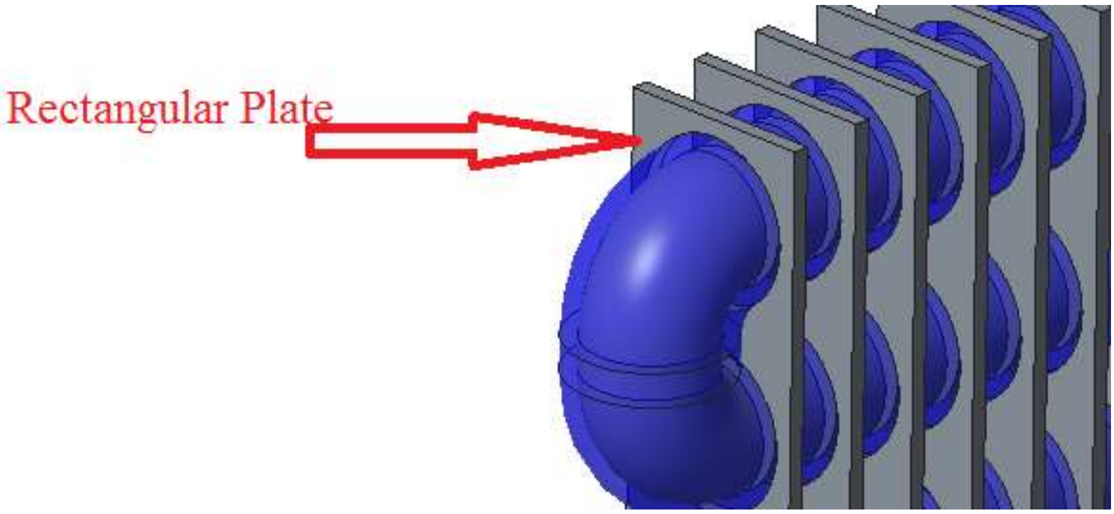




Analysis 3: CFD ANALYSIS OVER A PLATE, Fluid as ENGINE OIL

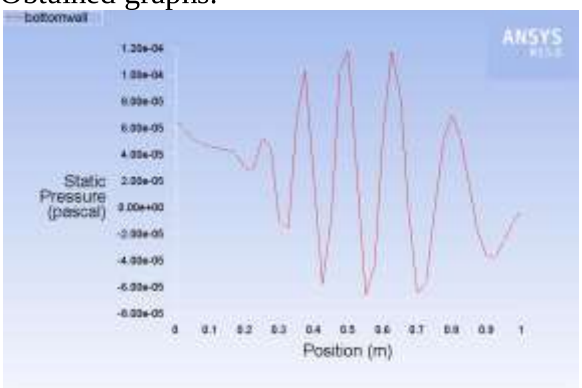
By considering fin as a rectangular plate and Engine Oil as fluid analyzed the variation of parameters either by keeping velocity as constant or temperature as constant.

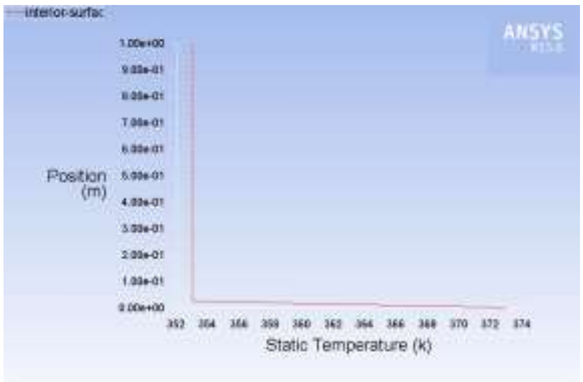
FLUID AS ENGINE OIL

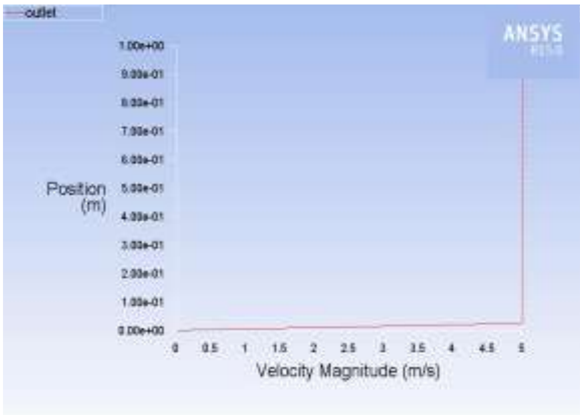


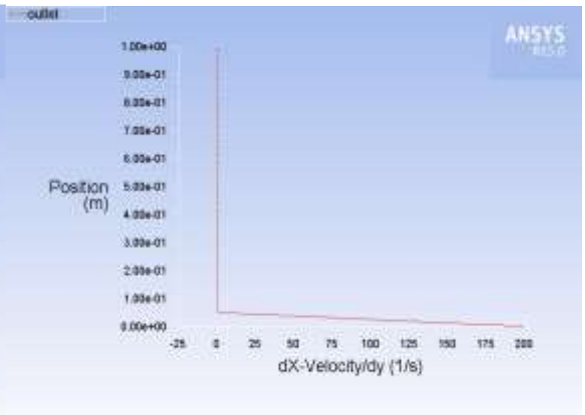
CASE 9: FLUID AS ENGINE OIL BY KEEPING TEMPERATURE CONSTANT & VARYING VELOCITY 5m/s:

Obtained graphs:



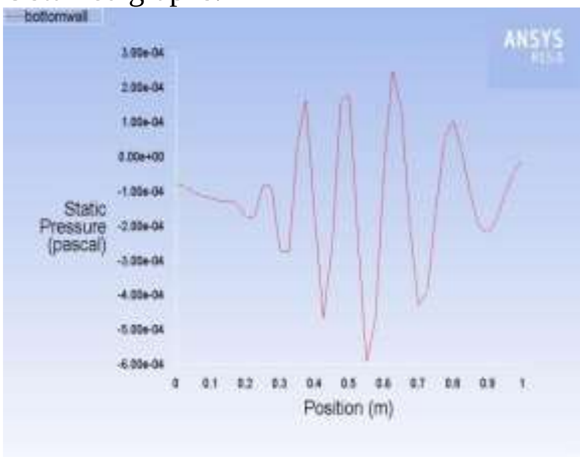


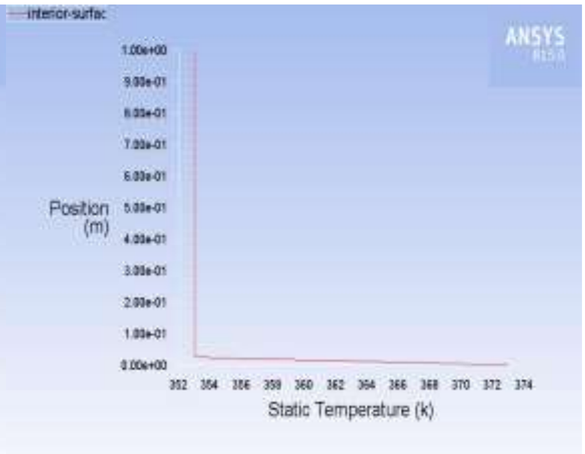


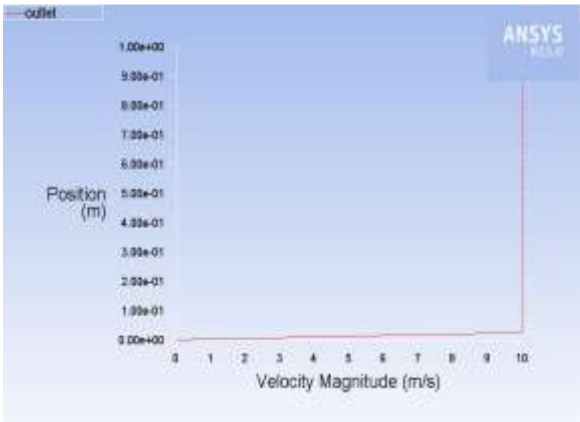


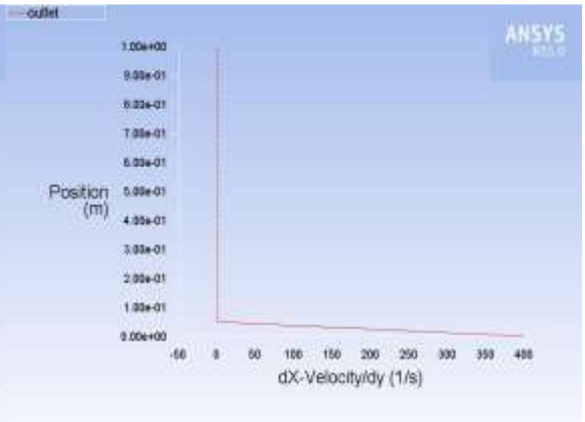
CASE 10: FLUID AS ENGINE OIL BY KEEPING TEMPERATURE CONSTANT & VARYING VELOCITY
10m/s:

Obtained graphs:



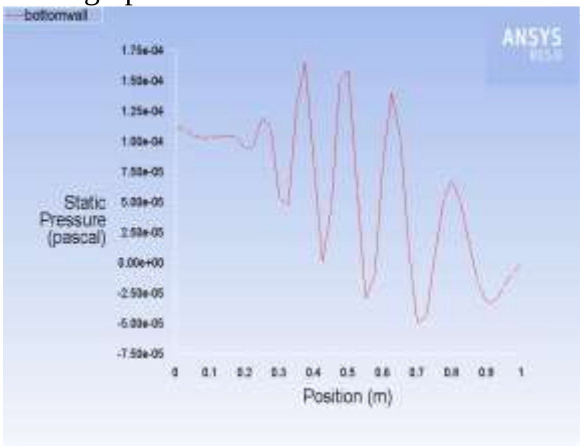


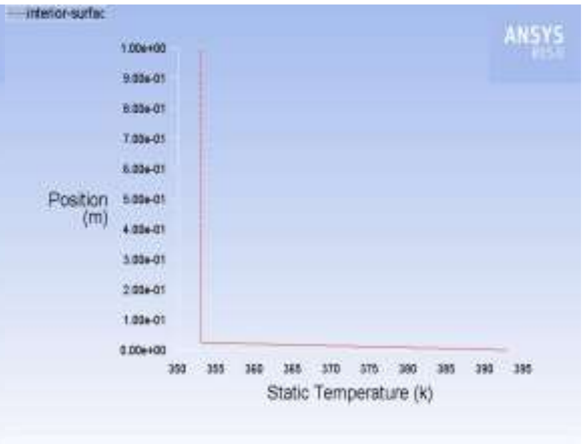


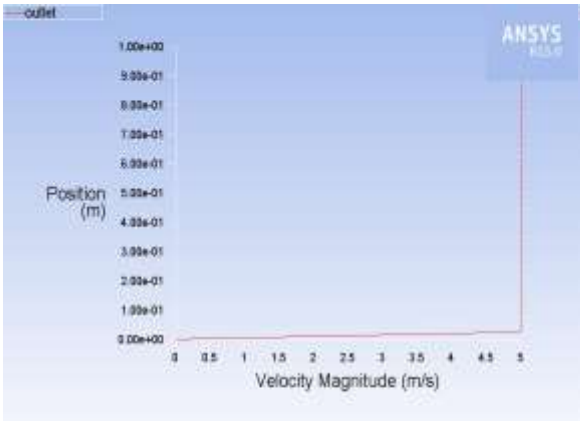


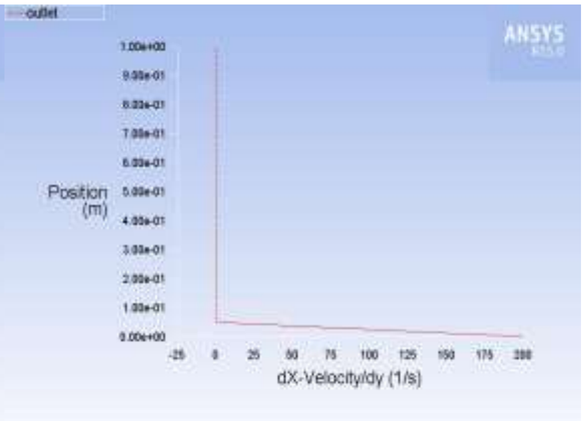
CASE 11: FLUID AS ENGINE OIL BY KEEPING VELOCITY AS CONSTANT & VARYING TEMPARATURE 393 k :

Obtained graphs:



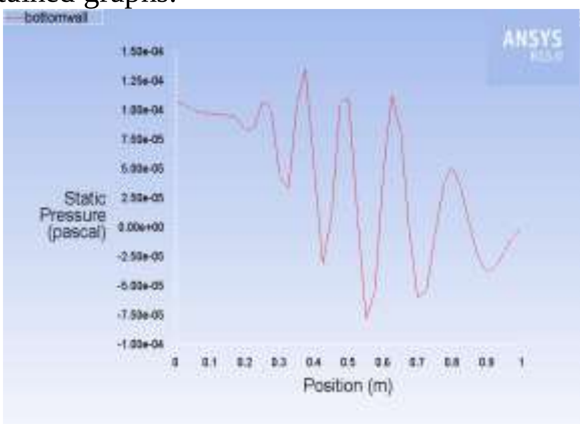


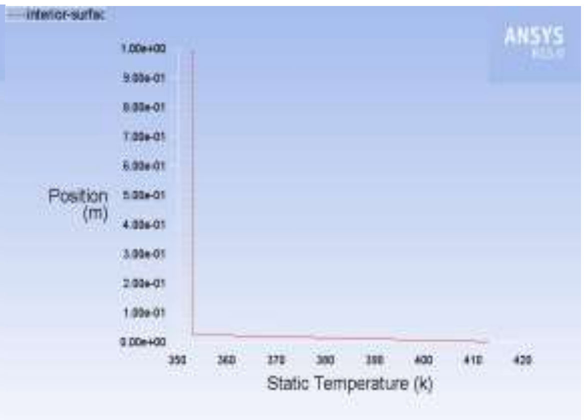


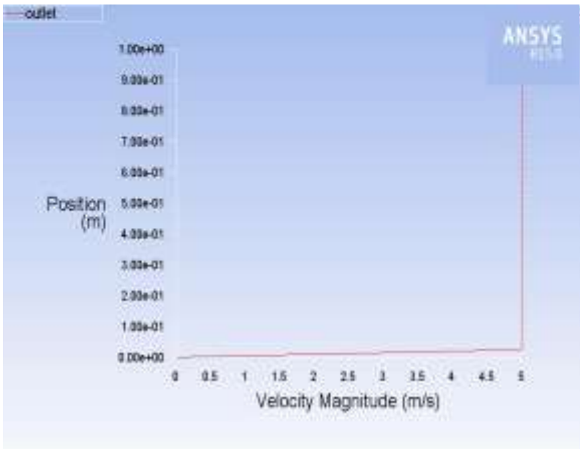


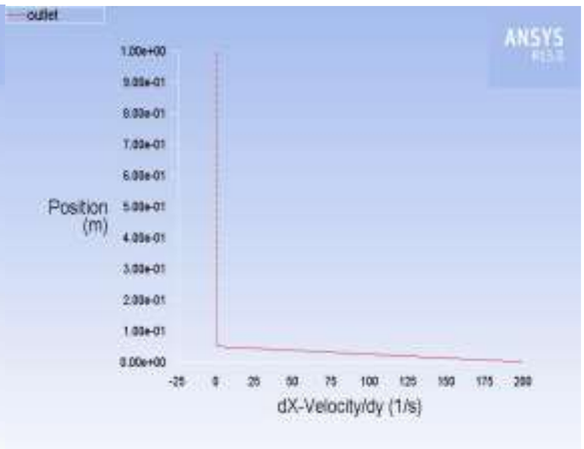
<u>CASE 12: FLUID AS ENGINE OIL BY KEEPING VELOCITY AS CONSTANT & VARYING TEMPERATURE 413 k :</u>

Obtained graphs:



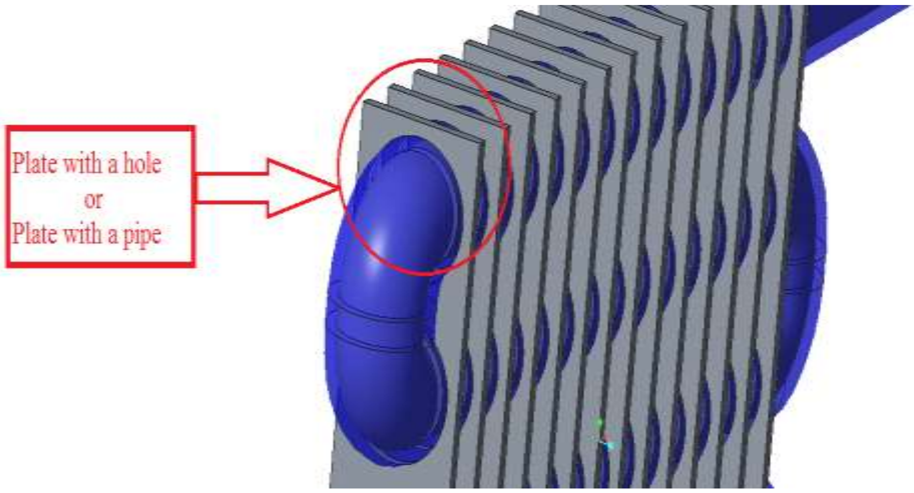






CFD ANALYSIS ON LAMINAR FLOW OVER A PIPE WITH FIN

Geometry:



MESHING :

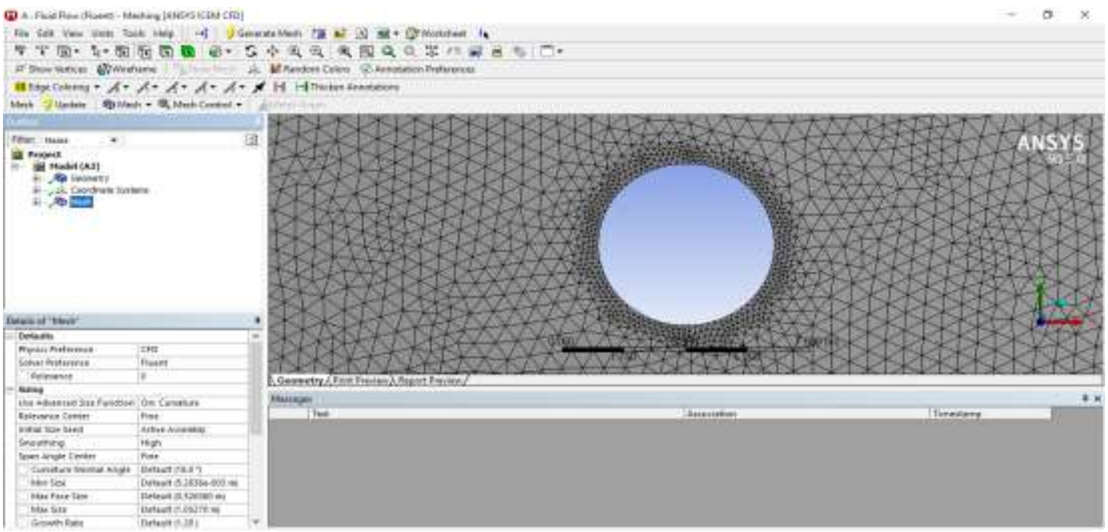
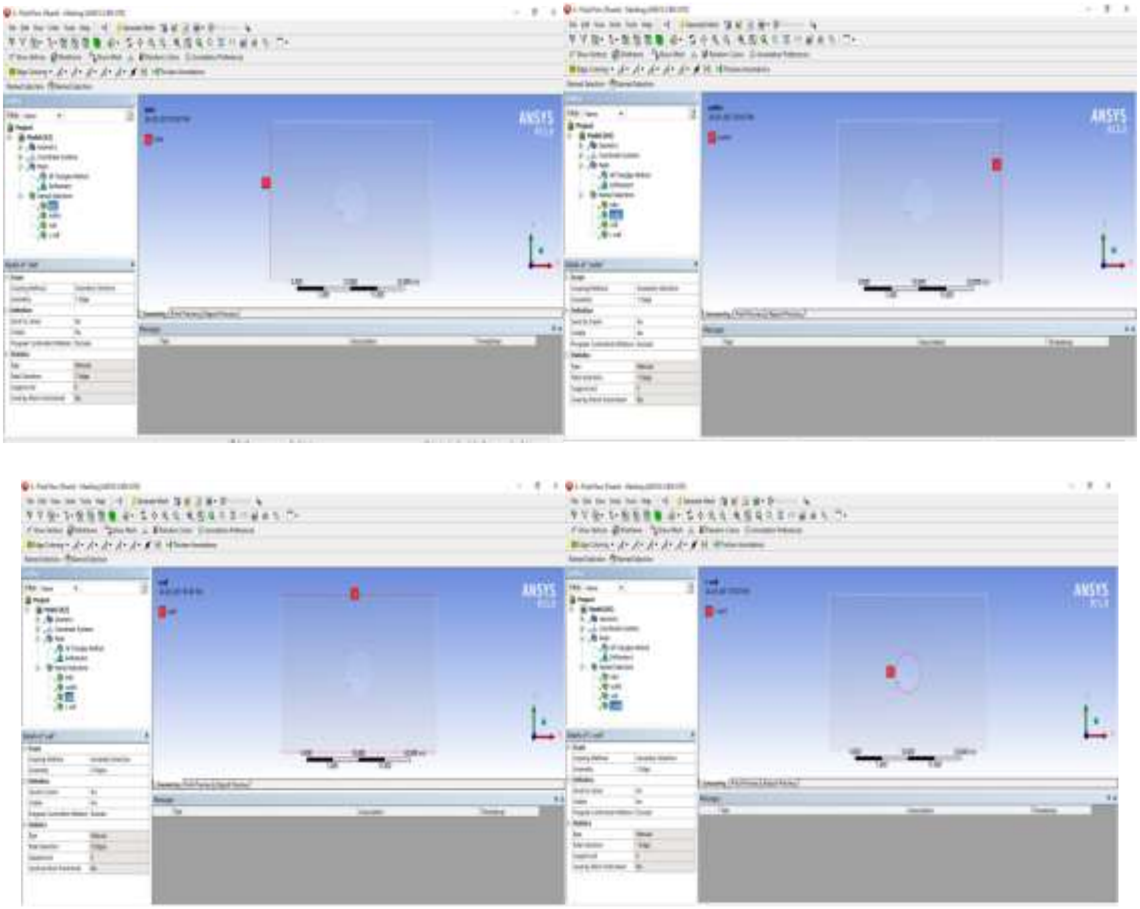


***Fig.3:**-Meshing of Plate with a Hole.*

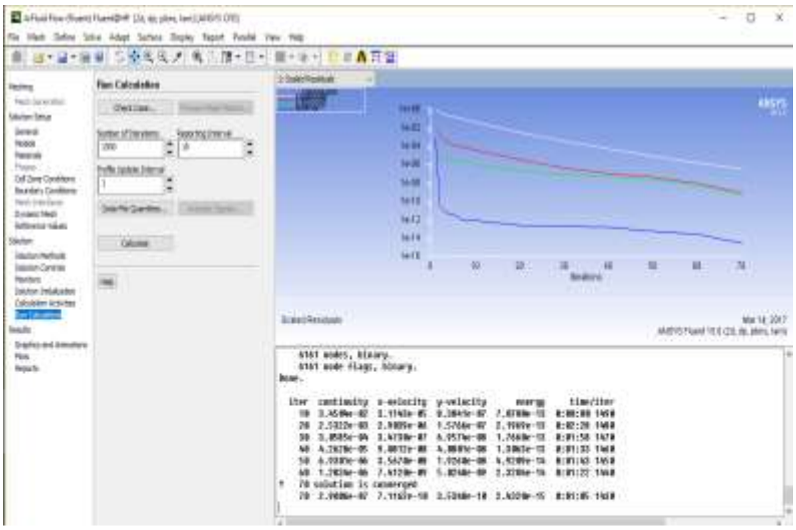
Strategy

Firstly we determine the flow in and flow out of the fluent.
Later, the wall and centre wall are determined. They are shown in the Figure 3.



Performing and Monitoring

By performing and monitoring the simulation process, the solution is calculated by the simulation and the graph is obtained.



CONCLUSIONS

In the present work, by considering fin as a rectangular plate and air, water and Engine oil as fluids velocity, pressure and temperature variations were analyzed. Similarly, the simulations are repeated for the plate with a hole. The condenser was created using CREO parameters software, and the designed condenser was confirmed using commercial Computational Fluid Dynamics software. CREO parameters 3.0 software is used to create a symmetric representation of the simplified geometry of a condenser [Pipe with Fins] for simulation purposes. CFD models or packages give outlines and data that may be used to forecast Heat exchanger designs which function well and are extensively used because of their ability to discover the optimum solutions.

FUTURE SCOPE

The current computational fluid dynamic analysis can be used for complex geometries that necessitate a thorough understanding of boundary conditions. Using 3D geometry, this work can be extended to a complete tube with fin condenser. Modern turbulence simulation methods, can also be added to the analysis. Working in CFD necessitates a strong foundation in both fluid mechanics and numerical analysis.

REFERENCES

1. Kanti, P. K., Karthika, U. P., Ali, S., Kumar, N. S., & Chandran, C. S. (2016). CFD Analysis of Shell and Tube Heat Exchanger. *International Journal of Engineering Research*, *5*(SP 6), 1249-1252.
2. Niphade, A., Chavan, H.(2016). Design, CFD Analysis of Shell & Tube Heat Exchanger CFD Analysis for Dairy Application. *International Journal of Advance Foundation And Research In Science & Engineering (IJAFRSE)2*(9).
3. Uppal, A., Kumar, V, Singh, C.(2014). Analysis of Heat Transfer Enhancement in a Heat Exchanger Using Various Baffle Arrangements. *IJRMET 4*(2).
4. Ali, S. M. Z. M. S., Krishna, K. M., Reddy, S. D. V. V. S., & Ali, S. R. S. M. (2015). Thermal analysis of double pipe heat exchanger by changing the materials using CFD. *International Journal of Engineering Trends and Technology*, *26*(2), 95-102.
5. Kumar, K, Srikesh, M.(2015). Computational Fluid Dynamics of Parallel Flow Heat Exchanger. *International Journal of Sciences: Basic and Applied Research (IJSBAR)* 24(5):111-120.
6. Johnson, J., Abdul Anzar, V. M., Shani, A., Harif Rahiman, P., Hashmi Hameed, T. S., & Nithin, V. S. (2015). CFD analysis of double pipe Heat Exchanger. *International Journal of Science, Engineering and Technology Research (IJSETR)*, *4*(5), 1283-1286.
7. Sarma, S., & Das, D. H. (2012). CFD Analysis of Shell and Tube Heat Exchanger using triangular fins for waste heat recovery processes. *IRACST–Engineering Science and Technology: An International Journal (ESTIJ), ISSN*, 2250-3498.