Numerical Investigation of Water Flow Dynamics in a Water Cooled Chiller Used in a Office Area

Jitul Baruah[1], Gangotri Borah[2]

[1]Associate Professor, Department of Mechanical Engineering, Assam Engineering College, Guwahati, Assam India

[2]Student, Department of Mechanical Engineering, Assam Engineering College, Guwahati, Assam, India

Abstract:

Water-cooled chillers play a vital function in a wide range of industrial and commercial applications, providing effective cooling solutions for air conditioning, refrigeration, and process cooling. The effective management of water flow dynamics inside these chillers' heat exchange systems is required for proper operation. The goal of this work is to do a detailed numerical analysis of water flow dynamics in water-cooled chillers in order to maximize their performance and energy efficiency. The key goals of this work are to model the fluid flow, heat transfer, and pressure drop characteristics in the chillers' heat exchangers, as well as to examine the impact of various design parameters on chillers performance, such as tube geometries, flow rates, and refrigerant qualities. The investigation is performed using computational fluid dynamics (CFD) simulations. The results were analyzed by using curves and bar diagrams. In essence, the goal of this work is to expand our understanding of water flow dynamics in water-cooled chillers by numerical analysis, ultimately leading to more efficient and sustainable cooling solutions in a variety of industrial and commercial contexts.

Keywords: water cooled chillers, CFD, air conditioning

1. Introduction:

Water-cooled chillers are vital components of heating, ventilation, and air conditioning (HVAC) systems used in a variety of applications ranging from vast industrial complexes to commercial buildings and data centers. By providing effective and dependable cooling, these machines play an important role in sustaining indoor comfort and preserving the quality of goods and equipment. Water-cooled chillers, in essence, are the workhours that keep the proper temperature and humidity levels in a variety of settings.

Water-cooled chillers are refrigeration systems designed to remove heat from aspecific space or process. They operate on the fundamental premise that heat is absorbed from the environment and then released elsewhere. This heat transfer is accomplished using a closed-loop system with water as a cooling medium. Water- cooled chillers, as opposed to air-cooled chillers, transmit heat to a separate water supply rather than directly into the ambient air.

Water-cooled chillers are distinguished by their capacity to transfer heat to a water supply, which can be a cooling tower, a river, or even a closed-loop system. Water-cooled chillers are useful for applications requiring substantial cooling capacities and precise temperature control due to their versatility. Furthermore, water-cooled systems are more energy-efficient than air-cooled counterparts, making them a preferred choice for cooling solutions in environments where long-term operational costs and sustainability are important consideration

The flow dynamics and cooling of water cooled chillers depends upon the following parameters

Coolant Circulation: Water-cooled chillers absorb heat from the space or process that requires chilling by circulating a water-based coolant (typically a blend of water and glycol). A closed-loop system is often used to pump the coolant. The coolant flow rate is an important characteristic that influences the chiller's cooling capacity and efficiency.

Evaporator: Warm water from the building or process passes over coils or tubes in thechiller's evaporator. Inside these coils, the refrigerant evaporates, collecting heat from the water and chilling it. To guarantee efficient heat transfer, proper water flow dynamics in the evaporator are critical.

Condenser performance: The condenser's water flow dynamics are equally significant. A separate water loop receives heat from the high-pressure, high-temperature refrigerant gas. Thecondenser's effective water flow ensures that the refrigerant is suitably cooled, allowing it to return to the evaporator and resume the cooling cycle

Pump and flow control device: Pumps are commonly used in chiller systems to circulate coolant through the evaporator and condenser. The flow rate of the water must be carefully managed to match the cooling demand, which is commonly accomplishedby modulating pump speeds with variable frequency drives (VFDs). Proper flow control ensures that the chiller runs efficiently and consistently provides consistent cooling performance.

Cooling Tower and Heat Exchanger: A cooling tower or heat exchanger is used in manywater-cooled chiller systems to disperse the heat absorbed by the coolant. Effective water flow dynamics in the cooling tower are critical for efficiently removing heat andmaintaining the coolant's optimum temperature.

Pipe and Distribution: The structure and design of the pipe system in a water-cooled chiller installation is also important for providing balanced water flow dynamics. Properly sized and engineered pipework reduces pressure drops and guarantees equal flow rates to all chiller components.

Temperature and pressure sensors: Temperature and pressure sensors are frequently used in modern water-cooled chillers to monitor and control water flow dynamics.

These sensors offer input to the chiller's control system, allowing for exact adjustmentof flow rates and temperatures in order to maintain peak performance. **[2]**

The Cooling load calculations have a critical part to determine water requirements, which acts as the foundation for the overall optimization of the building's cooling system. This procedure entails a thorough examination of the building's thermal requirements,which includes a thorough examination of the many components that influence heat acquisition and loss [**5**]**.** The following main points will help you understand the significance of cooling load calculations:

Accurate Sizing of the Cooling System, avoiding overdesign and underperformance, Optimization of energy utilization, building specific factors, Guiding system design and implementation.

The precise cooling load calculations are critical to the project's success because they provide a data-driven foundation for designing and optimizing the building's cooling system. This method helps to eliminate inefficiencies caused by overdesign or underperformance, resulting in a properly sized and energy-efficient cooling solution adapted to the individual requirements of the building.

The numerical analysis of water flow dynamics in water-cooled chillers is a thorough investigation that employs computational tools and mathematical modelling to examine and optimize the behaviour of water as it circulates through the various components of a water-cooled chillers system. This analysis is critical forcomprehending how water flow dynamics influence the performance, efficiency, and dependability of water-cooled chillers. The following are the main components of such an analysis:

Computational Fluid Dynamics (CFD): CFD is a powerful numerical method used to simulate and study the behavior of fluids within the chiller's components, including water. Engineers and academics can use it to predict and visualize water flow, temperature distribution, and pressure fluctuations across the system.

Flow Path Modelling: Engineers build mathematical models of the water flow path within the chiller, which includes the evaporator, condenser, pipes, pumps, and cooling towers. These models aid in predicting how the dynamics of water flow affectheat transfer, energy consumption, and cooling capacity.

Heat Transfer study: The numerical study comprises evaluations of the chiller's heat transfer properties. It takes into account how well the water absorbs heat in the evaporator and releases it in the condenser, as well as the effect of flow rates and temperature differentials.

Pressure Drop Analysis: It is critical to understand pressure losses within the chiller's components. Numerical simulations aid in the identification of pressure drop sites andtheir effects on flow dynamics and pump requirements.

Efficiency and Energy Analysis: Numerical models allow for the assessment of chillerefficiency and energy consumption under various situations, assisting in the identification of potential energy-saving options.

Validation Using Experimental Data: Numerical findings are frequently tested with real-world experimental data to confirm the simulation's accuracy and reliability.

The primary objectives of a numerical analysis of water flow dynamics in water-cooled chillers are as follows:

Improve energy efficiency by optimizing the design and operation of chillers.

Predict and avoid problems caused by flow imbalances, pressure decreases, or insufficient heat transmission are an important part of its operation, determining its efficiency, performance, and overall efficacy in cooling applications. The circulation of water via numerous components and channels within the chillier characterizes Improve the dependability and performance of chiller systems in a variety of operational environments.

Improve the long-term viability of cooling solutions by reducing energy usage and environmental effect.

To summarize, numerical study of water flow dynamics in water-cooled chillers employs computational modelling and simulation approaches to acquire a thorough understanding of how water circulates and interacts within the chiller system. This knowledge is crucial for optimizing chiller design and operation, lowering energy costs, and assuring reliable and effective cooling in a variety of applications **[1]**

Apart from these computational study maintenance plays a vital role: Regular maintenance, including filter replacements,cleaning, and inspections, is crucial for HVAC systems to operate efficiently and maintain optimal performance. Additionally, advancements in technology, such as Building Management Systems (BMS) and predictive maintenance, enable proactive monitoring and optimization of HVAC systems.

Finally, it can be said that HVAC systems play a vital role in creating a comfortable and healthy indoor environment. By regulating temperature, humidity, and air quality, these systems enhance the well-being and productivity of building occupants. Energy efficiency, proper design, regular maintenance, and technological advancements are key considerations in optimizing HVAC system performance and achieving sustainable and cost-effective operation **[14]**

2. Methodology:

The methodology for the work involves a systematic approach to accurately assess the thermal requirements which is found to be 12 TR as calculated previously and analyze the flow dynamics of water within the chiller system. The following steps outline the key components of the methodology.

2.1. Cooling Space: An architectural plan of a floor of an office building has to be designed with 4000 sq.ft area, which contains an office room, a cafeteria, a printout shop and a store room and a toilet with some specific dimension. For cooling the space three numbers of 2TR and two numbers of 3 TR AHU is used

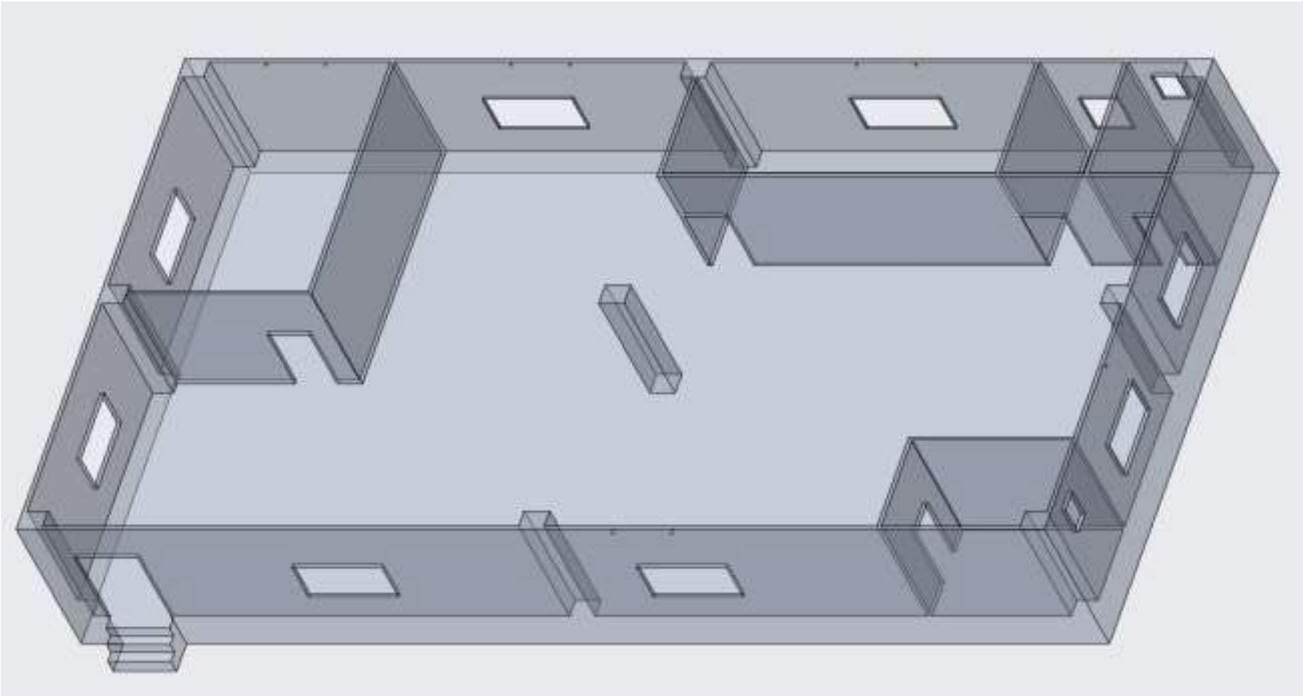

Figure 1: 3-D design of a office building with 4000 sq. ft

2.3 Water requirement Calculations:

$$TR = \frac{GPM(US)\times 60\times Sp\ heat\ of\ water\times \Delta t(°F)}{12000}$$

1 Gallon per minute (GPM) (US) = 8.33lbs

$$TR = \frac{GPM\times 8.33\times 60\times Sp\ heat\ of\ water\times \Delta t(°F)}{12000}$$

Let, Chiller outlet temperature = 6°C = 42.8°FChiller inlet temperature = 12°C = 53.6°F

Δt = (53.6-42.8) °F = 10.8 °F

$$TR = \frac{GPM\times \Delta t(°F)}{24}$$

$$GPM = \frac{TR \times 24}{\Delta t(°F)}$$

$$GPM = \frac{12 \times 24}{10.8(°F)} = 26.667 \text{ GPM}$$

Qwater= 26.667 GPM (gallons per minute)

1 GPM = 3.78541 Litres per minute 26.667 GPM = 100.9443 Litres per minute

The required water flow rate of the chiller, Qwater= 100.9443 Litres per minute

= 1.682405 Lt/sec.

= 0.0016824 m^3/sec.

We know that,

Q = A.v

A= Cross-sectional area of inlet pipev= inlet velocity of water

$$v = Q/A = \frac{0.0016824}{\pi(0.02)^2} \text{ m/sec} = 1.3388 \text{ m/sec}$$

For 2 TR AHU,

$$TR = \frac{GPM \times \Delta t(°F)}{24}$$

$$GPM = \frac{TR \times 24}{\Delta t(°F)}$$

$$GPM = \frac{2 \times 24}{10.8(°F)} = 4.4444 \text{ GPM}$$

Qwater(2) = 4.4444 GPM (gallons per minute)

= 0.280398 Lt/sec.

For 3 TR AHU,

$$TR = \frac{GPM \times \Delta t(°F)}{24}$$

$$GPM = \frac{TR \times 24}{\Delta t(°F)}$$

$$GPM = \frac{3 \times 24}{10.8(°F)} = 6.6667 \text{ GPM}$$

Qwater (3) = 6.6667 GPM (gallons per minute)

= 0.420603 liter/sec

2.4 Pipe Size determination: To Determine the Pipe size ASHRAE's Chilled water pipe size is used
For Branch Pipe: For 2TR AHU, we calculated that the Flow rate as 4.4444 GPM Hence, from the table we can get, pipe diameter = 0.75 in or 20 mm. For 3TR AHU, we calculated that the Flow rate as 6.6667 GPM Hence, from the table we can get, pipe diameter = 1 in or 25 mm. Here, let us consider the Velocity of water will be 0.75 m/s.
For Main Pipe: The main pipe diameter is found to be 1.5 inch or 40 mm.

2.5. Design of the outlet and inlet pipes:

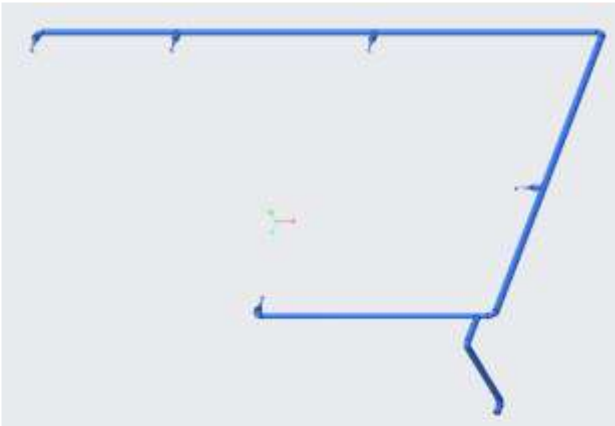

Figure 2: Inlet Pipe

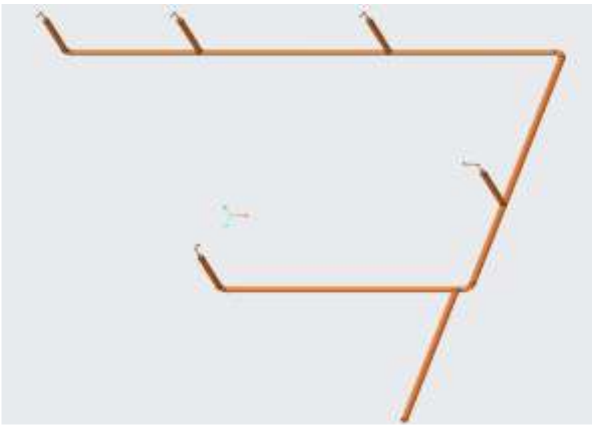

Figure 3: Outlet Pipe

Length of the total inlet pipe =48586.1 mm and total volume of inlet pipe = 61055093.93 mm^3 = 2.156 ft^3
Length of the total outlet pipe =68966.04 mm and total volume of inlet pipe =86665281.84 mm^3 = 3.06 ft^3. Hence, let us consider total volume of the tank required is 6 ft^3 i.e. 0.17 m^3. therefore approximate 170 liters water required to achieve the desired temperature.

2.6 Experimental Setup: The Experimental setup is now designed in Ansys Design Modeler and then simulated using Ansys Fluent 2022 R1 using student version:
Using Ansys Fluent to analyses the water flow of the inlet and outlet pipe of the water-cooled chiller requires a methodical approach. Here is a general outline of the steps that should typically follow:

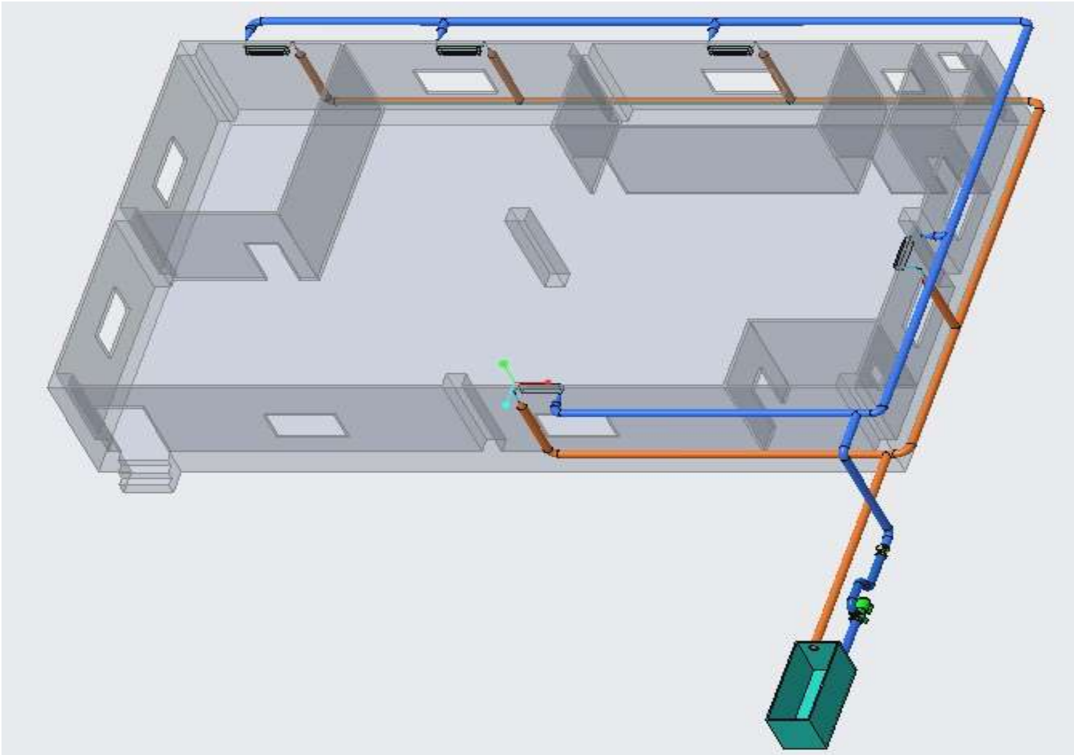

Figure 4: 3D model of the office building with air conditioner, inlet and outlet pipe with pump and chiller

2.6.1 Problem Setup:

In the problem setup, the geometry of inlet and outlet pipe has to import in the Ansys design modeler. In this geometry, the main pipeline with inner diameter 40mm and three branch pipes has the inner diameter as 20mm and two branch pipes has inner diameter 25mm.

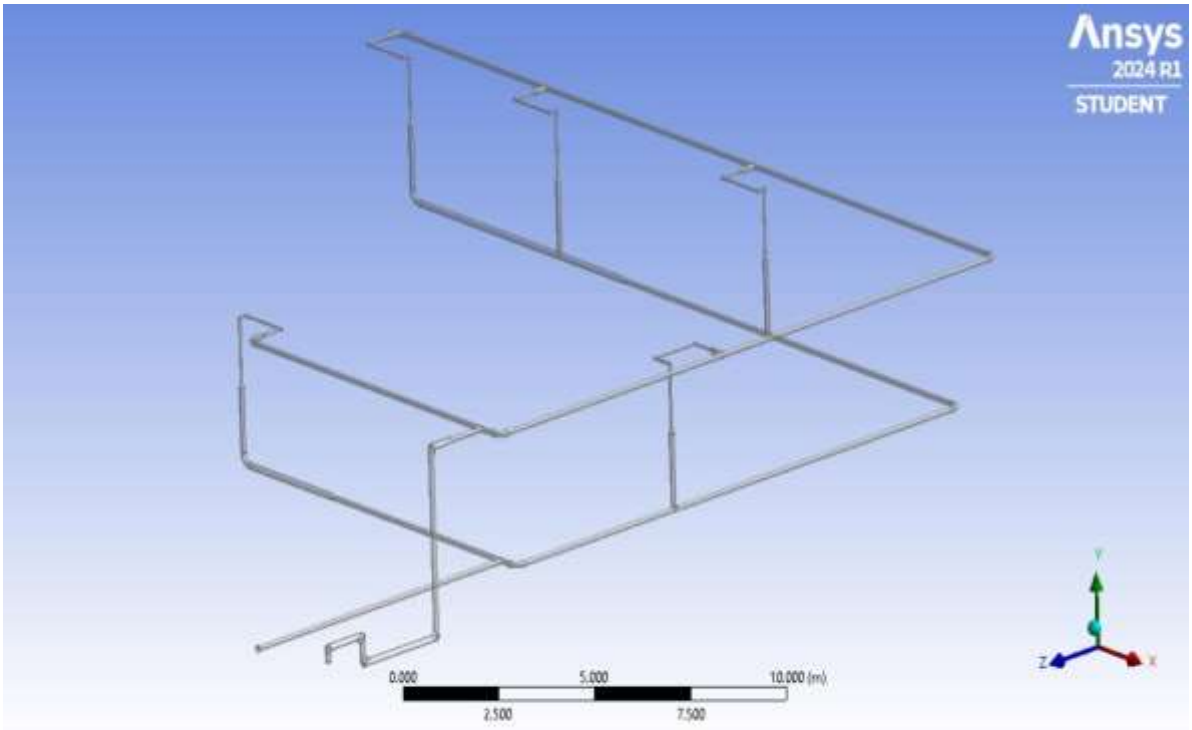

Figure 5:Geometry of inlet and outlet pipe assembly

2.6.2 Mesh Generation:

In mesh generation, a mesh for the pipeline geometry has to be created. The mesh should be thin enough to capture flow details while remaining computationally efficient.

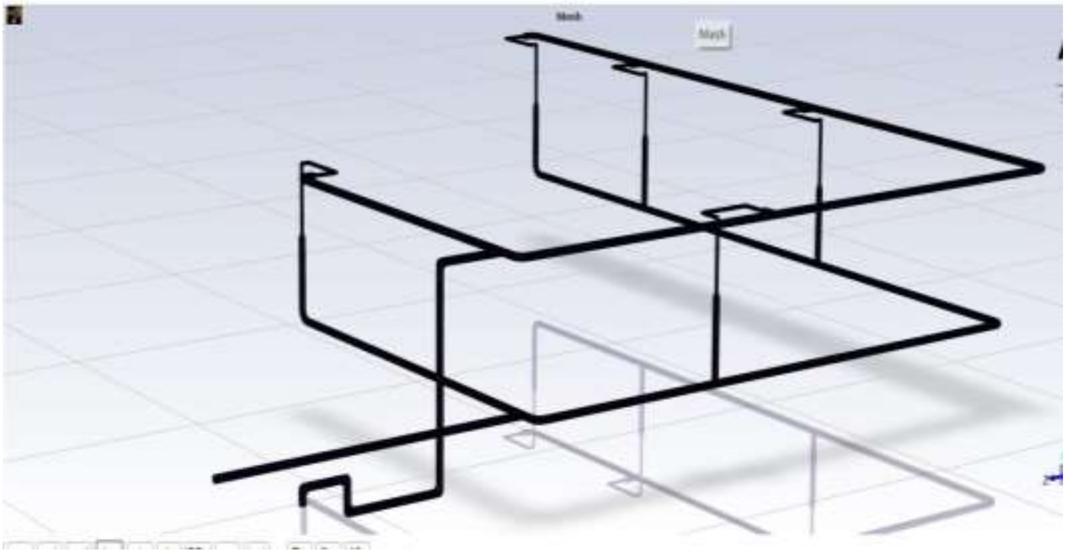

Figure 6: Mesh generation of pipe assembly

2.6.3 General Processing: Pressure based solver type is used for the simulation with absolute velocity formation. The time is taken as steady. Gravity of 9.81m/s^2 is applied on y direction.

2.6.4 Boundary Conditions: The boundary conditions for each case within the parameter study have to be set. This could include modifying the input velocity, temperature, or other pertinent parameters.This includes, The input velocity as 1.3388 m/sec, Chilled water temperature at inlet is 6◦c Material used in pipe is cast iron with conductivity 52 W/mk. For the calculation of the heat loss in pipe, Let, Ambient temperature =35◦c Temperature inside the room = 23◦c Convective heat transfer coefficient of air = 20 W/m^{2}k

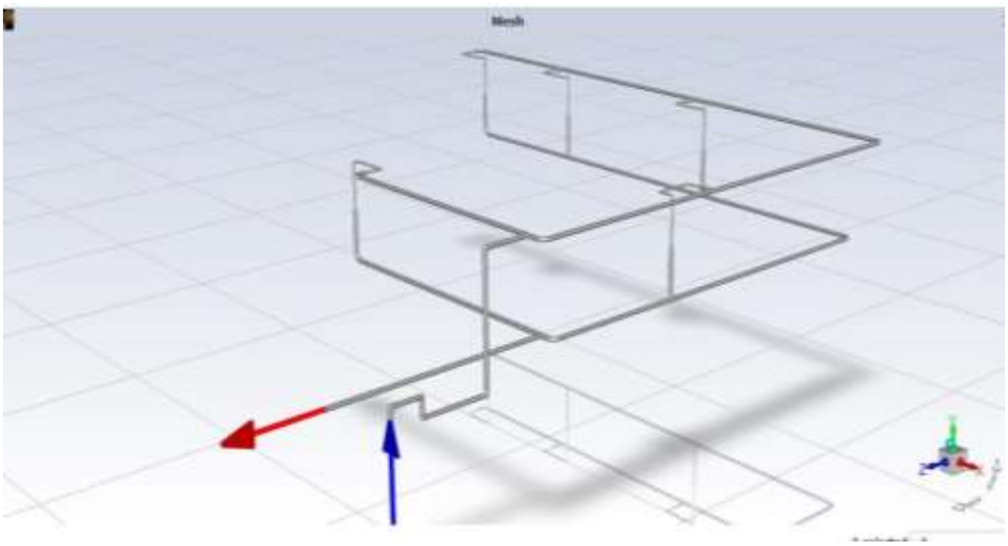

Figure 7: Inlet and outlet direction of pipe assembly

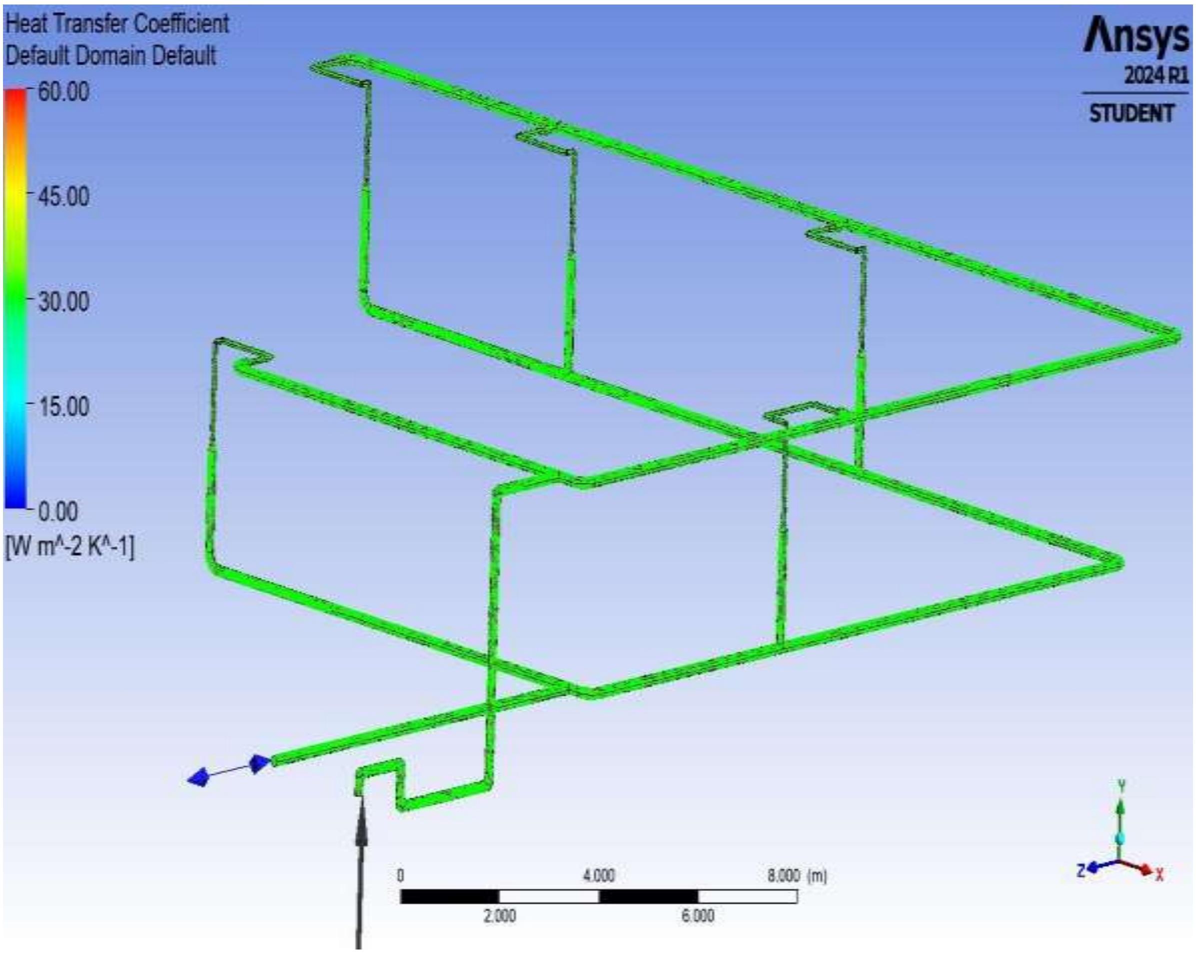

Figure 8: Boundary Conditions of heat transfer Coefficient of air

2.6.5 Solution Methods: The pressure-based segregated algorithm solves the momentum equation and pressure correction equations separately. Slow convergence is the result of this semi-implicit solution approach.

2. 6.6 Initialization and running of calculation: Initialization is performed with computing starting from velocity inlet andthen the calculation is run. A total no of 50 iterations is considered keeping the computing time in mind

3. Result and Discussion

3.1 Cooling load: The total cooling load for the 4000 sqft office floor is a critical parameter inthe design of the HVAC (Heating, Ventilation, and Air Conditioning) system, specifically in this case, the selection of a chiller system. The cooling load represents the amount of heat that needs to be removed from the space to maintain a comfortableand controlled temperature.

Table 1: Total Load Capacities

Loads	Total Sensible load (BTU/hr)	Total Latent load (BTU/hr)	Total Load (BTU/hr)
Wall Load	31269.888		31269.888
Roof Load	23621.84		23621.84
Door Load	1593.888192		1593.888192
Window Load (conduction)	2697.8184		2697.8184

Window Load (Solar)	7963.23		7963.23
Lighting Load	15464.2488		15464.2488
People Load	18400	15200	33600
Ventilation Load	3391.07	13310	16701.07
Infiltration	8314.89		8314.89
Miscellaneous Load	471		471
			141697.8734

The total cooling load for the 4000 sqft office floor was determined to be 141697.8734 BTU/hr.
The resulting cooling load, expressed in BTU/hr, provides a clear understanding of the thermal demands of the space. This value is used as a basis for selecting a chiller system with a capacity sufficient to handle the calculated cooling load, ensuring efficient and reliable cooling performance.

3.2 Water requirement: The required amount of water for the water-cooled chiller system is a crucial parameter for designing the water circulation system that facilitates the heat rejection process. This involves taking the heat absorbed from the building by the chiller and dissipating it into the environment through the water-cooling process. The water requirement, expressed in gallons per minute (GPM), provides thenecessary information for designing the water system.
The water flow required for the main pipe is 1.682405 Lt/sec.
The total water requirement for the 4000 sqft office floor was determined to be 170 liters. The volume of the tank will be 6 ft^3 i.e. 0.17 m^3.

3.3 Results of CFD Simulation: For result and analysis part, the assembly should be divided into various sections for the analysis of the results.

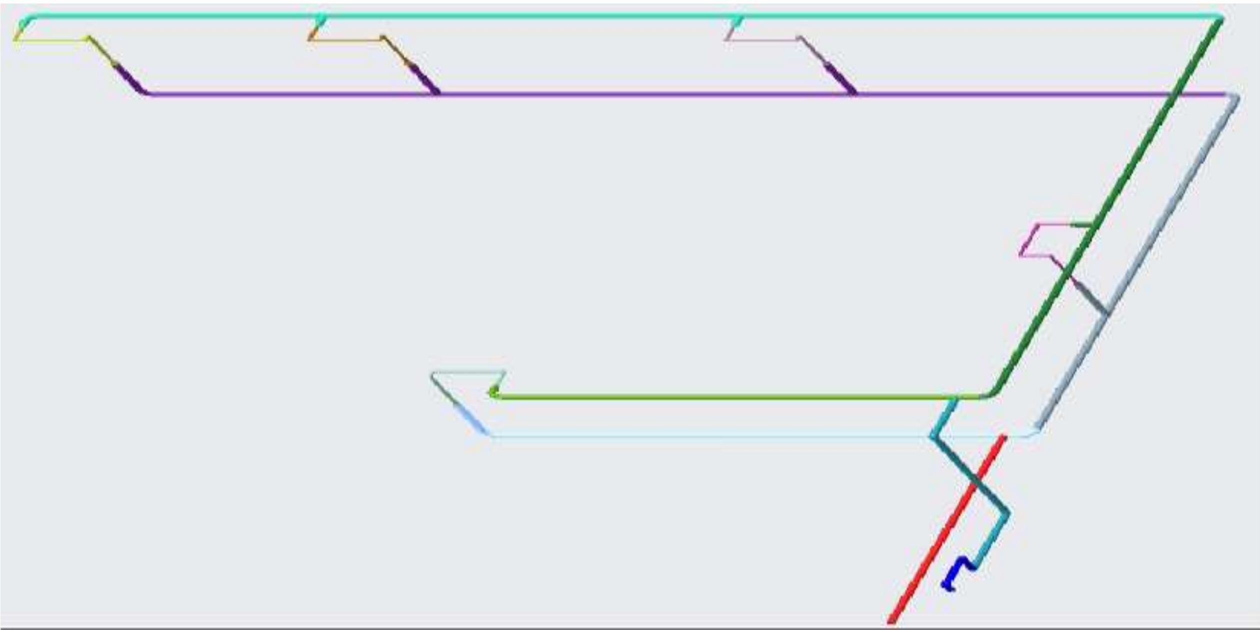

Figure 9: Different section of pipe assembly

3.3.1 Pressure and Flow Analysis*: Pressure analysis in a CFD study of water-cooled chiller pipes involves understanding the intricate variations in pressure throughout the system and identifying the sources of pressure losses. To examine the pressure dynamics and fluidflow patterns to identify potential* *areas of inefficiency or flow restrictions within the water circulation system turbulence levels. Optimize pipe diameters, layouts, and pump configurations to ensure uniform and efficient water flow and to conduct CFD simulations to visualize pressure drops, flow velocities. By generating a detailed map of pressure distribution along the pipes, the analysis reveals how pressure varies from one section to another, highlighting areas of high and low pressure. This pressure distribution map is crucial for identifying potential issues such as blockages, leaks, or design inefficiencies. Frictional losses, which occur due to the interaction between thefluid and the pipe walls, are quantified using the Darcy-Weisbach equation, considering factors like fluid velocity, pipe diameter, and surface roughness. These losses are spread throughout the length of the pipe and are a significant contributor to the overall pressure drop. Additionally, minor losses arise from flow disturbances caused by bends, fittings, valves, and other geometric features. These losses are typically calculated using empirical coefficients that represent the energy dissipation at these points. Accurate quantification of both frictional and minor losses is essential for pump sizing, ensuring that the pump can provide sufficient pressure to maintainthe desired flow rate without being oversized, which would lead to energy inefficiency, or undersized, which would compromise system performance. Moreover,understanding these pressure losses allows for optimization of the chiller design by minimizing unnecessary losses through design improvements such as smoother bends or optimized pipe diameters. This not only improves the chiller's durability andperformance, but also increases its energy efficiency, lowering running costs and contributing to a more sustainable cooling solution.*

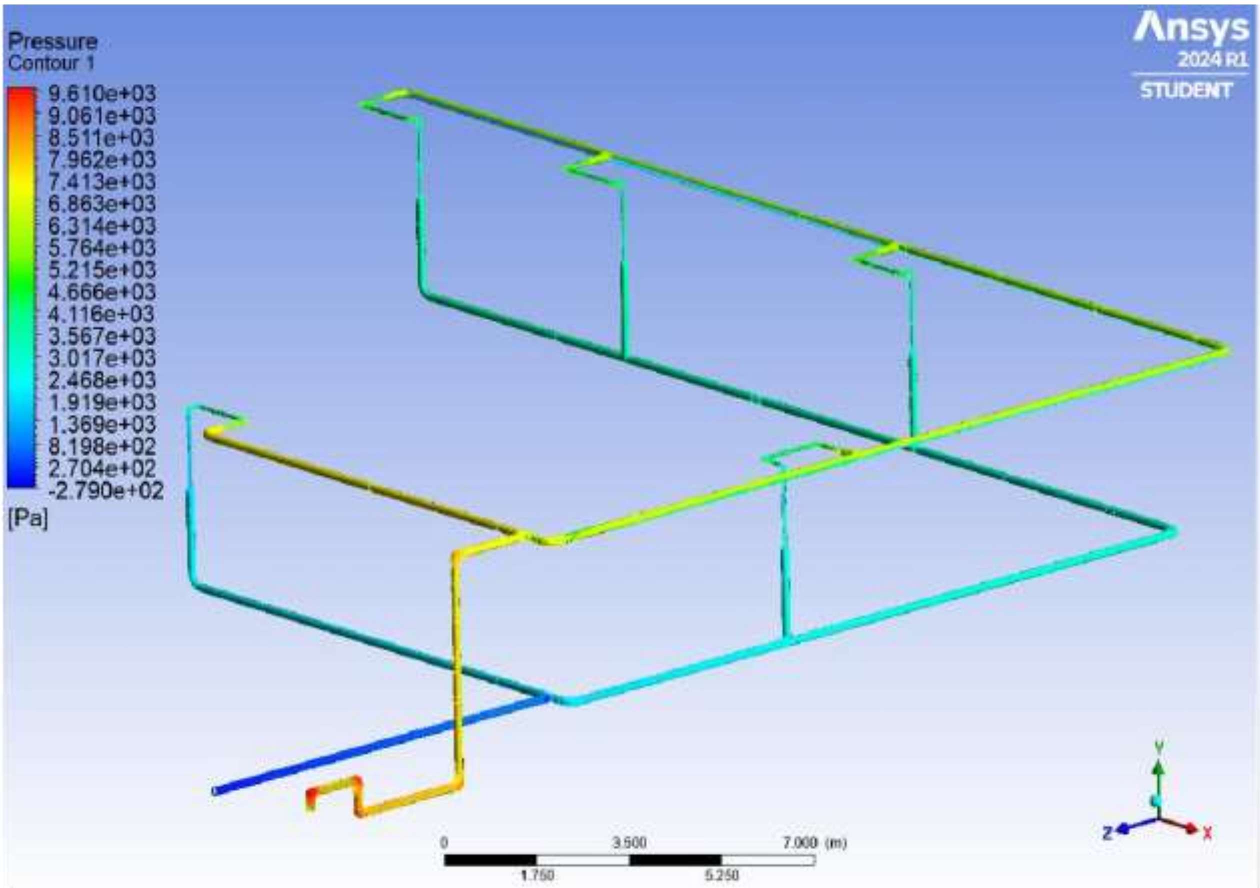

Figure 10: Pressure contour of the pipe assembly

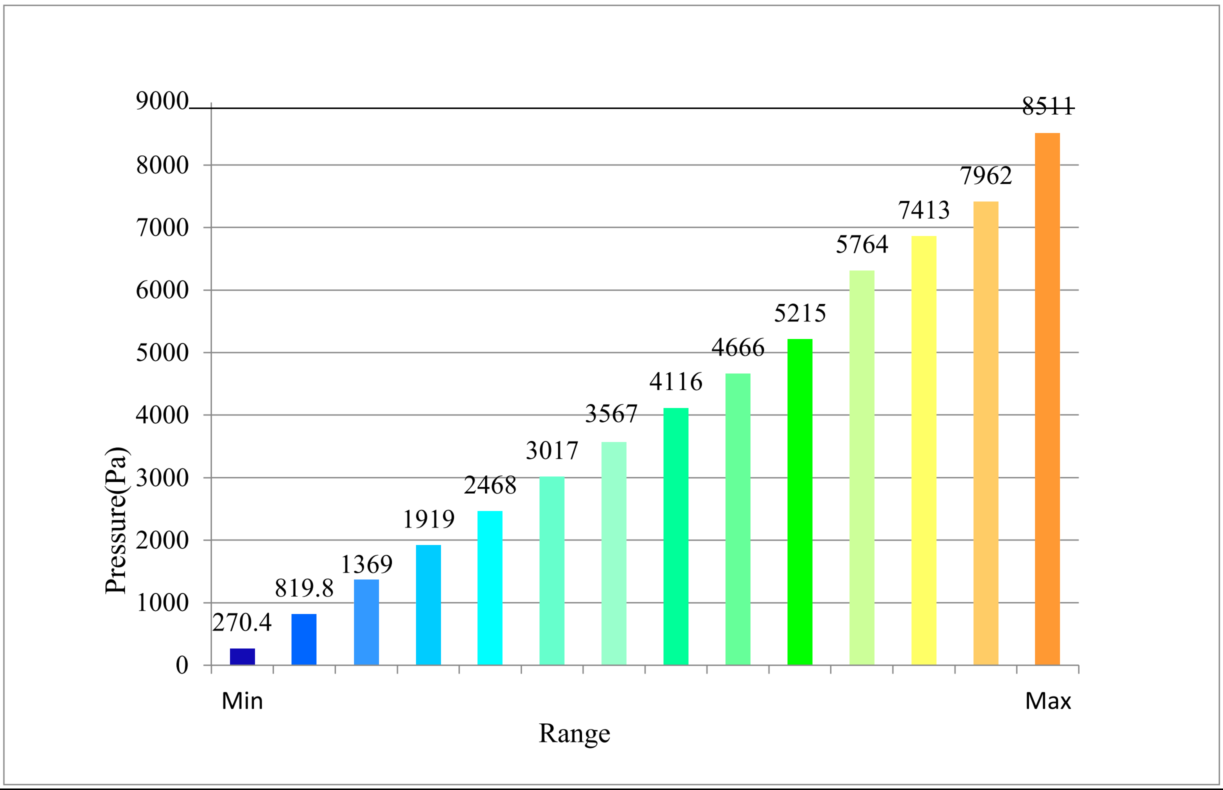

Figure 11: Variation of pressure in different pipe sections

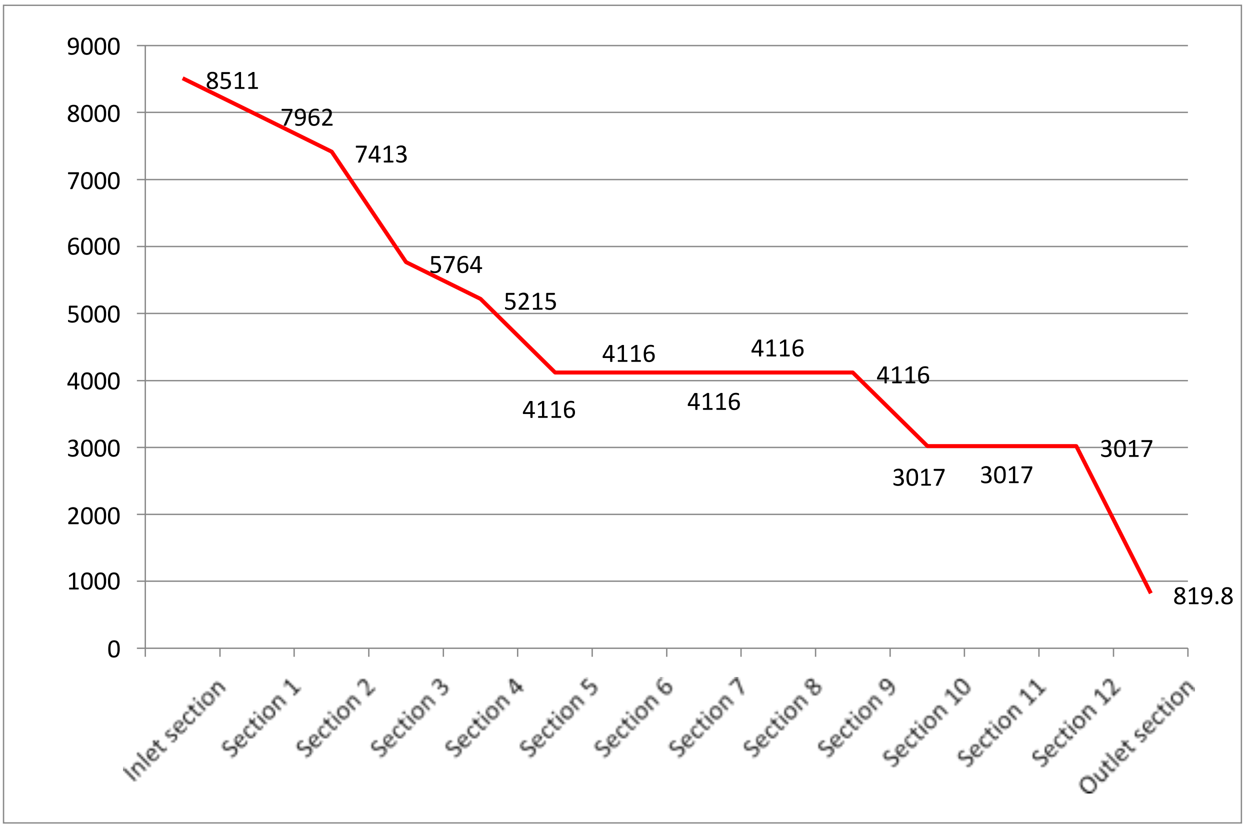

Figure 12: Variation of Pressure in bar diagram throughout the pipe

In figure 11 and figure 12; pressure gradually decreases throughout the pipe because of various factors like velocity, frictions; in the inlet section the pressure will be 8511 Pa, as the fluid enters from the inlet section to the SECTION 1 of the pipe assembly the pressure of the water will decrease to 7962 Pa as the length increases there will be a friction, from SECTION 1 to SECTION 2 the pressure of the liquid decreases to 7413 Pa, from SECTION 2 to SECTION 3 the pressure of the water will be 5764 Pa, from SECTION 3 to SECTION 4 the pressure of the water will be 5215 Pa, after the SECTION 4 water goes through SECTION 5, SECTION 6,SECTION 7, SECTION 8, SECTION 9 which contains the air handling units with smaller diameter i.e. 20 mm than the main pipe diameter 40 mm in these sections the pressure shows 4116 Pa, from the air handling units the water flows through the SECTION 10, SECTION 11, SECTION 12 respectively and the pressure in this sections will be3017 Pa, then the water flows to the outlet section with pressure 819.8 Pa

3.3.2 Velocity Analysis: Flow distribution and velocity profiles obtained from CFD analysis are fundamental to understanding the internal dynamics of water-cooled chiller pipes, ensuring efficient system performance and optimal heat exchange. The flow distribution analysis reveals how uniformly water flows through the pipes, which is critical for maintaining consistent cooling and avoiding the formation of hotspots that can lead to inefficient heat transfer and potential system failures. Uneven flow distribution can cause certain sections of the chiller to experience higher thermalloads, thereby reducing the overall efficiency and lifespan of the system. In additionto assessing flow uniformity, the CFD analysis provides detailed velocity profiles at various cross-sections of the pipes. These profiles show the speed and direction of water movement at various points along the pipe, highlighting areas where flow patterns diverge from the expected uniformity. Specifically, velocity profiles candetect locations of strong turbulence, which can increase frictional losses and lower energy efficiency, as well as areas of recirculation or stagnation. Such zones are hazardous because they can cause localized overheating and limit heat transmission efficiency. Furthermore, identifying low flow locations is critical since these places may be prone to sedimentation or scaling, reducing flow and heat exchange. Engineers can use these velocity profiles to identify design flaws or operationalconcerns, such as sharp bends or incorrect pipe diameters, which disturb smooth flow. This detailed understanding allows for targeted design improvements and operational adjustments, such as changing pipe geometries or flow rates, to ensure a more uniform flow distribution and optimal velocity profiles, thereby improving the overall performance

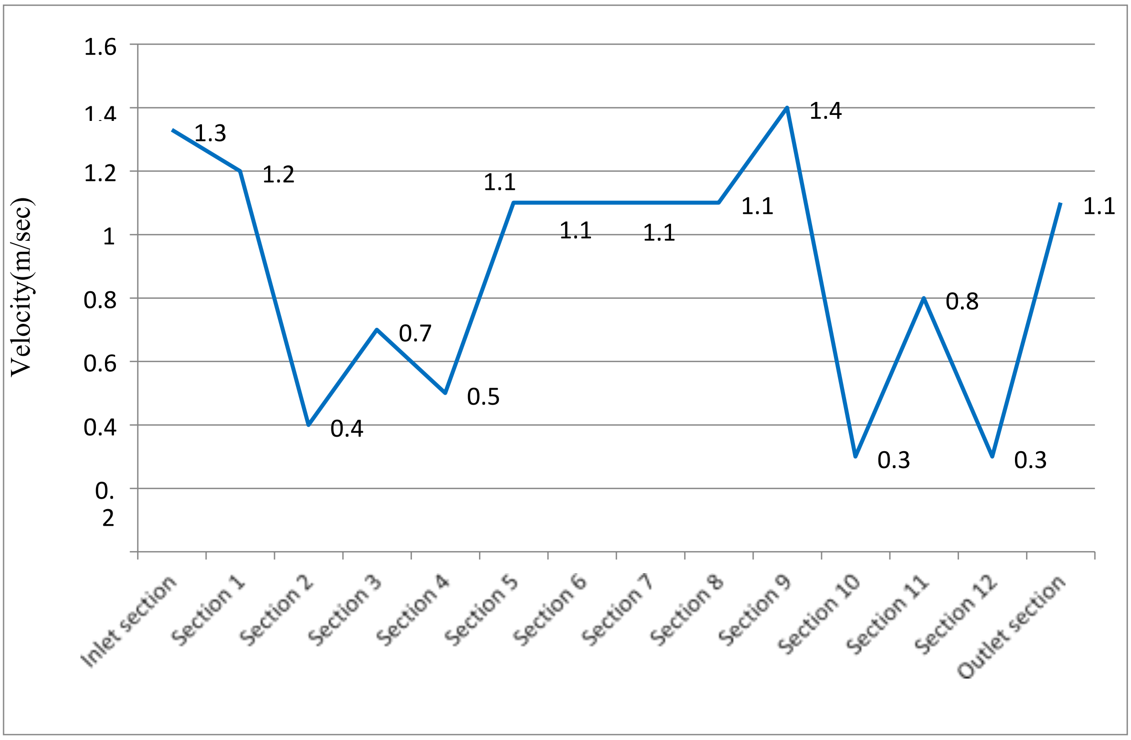

Figure 13: Velocity variation inside the pipe throughout the section

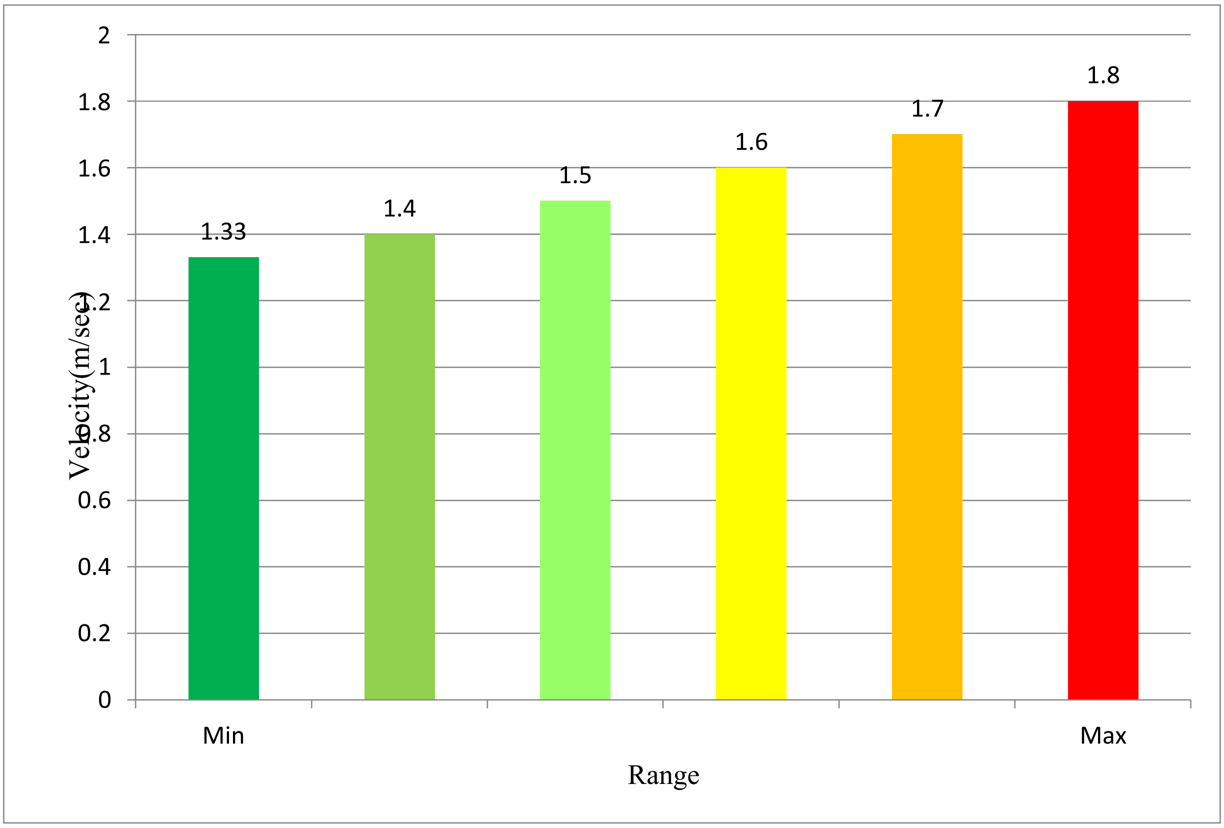

Figure 14:Velocity Variation inside the pipe in bar diagram

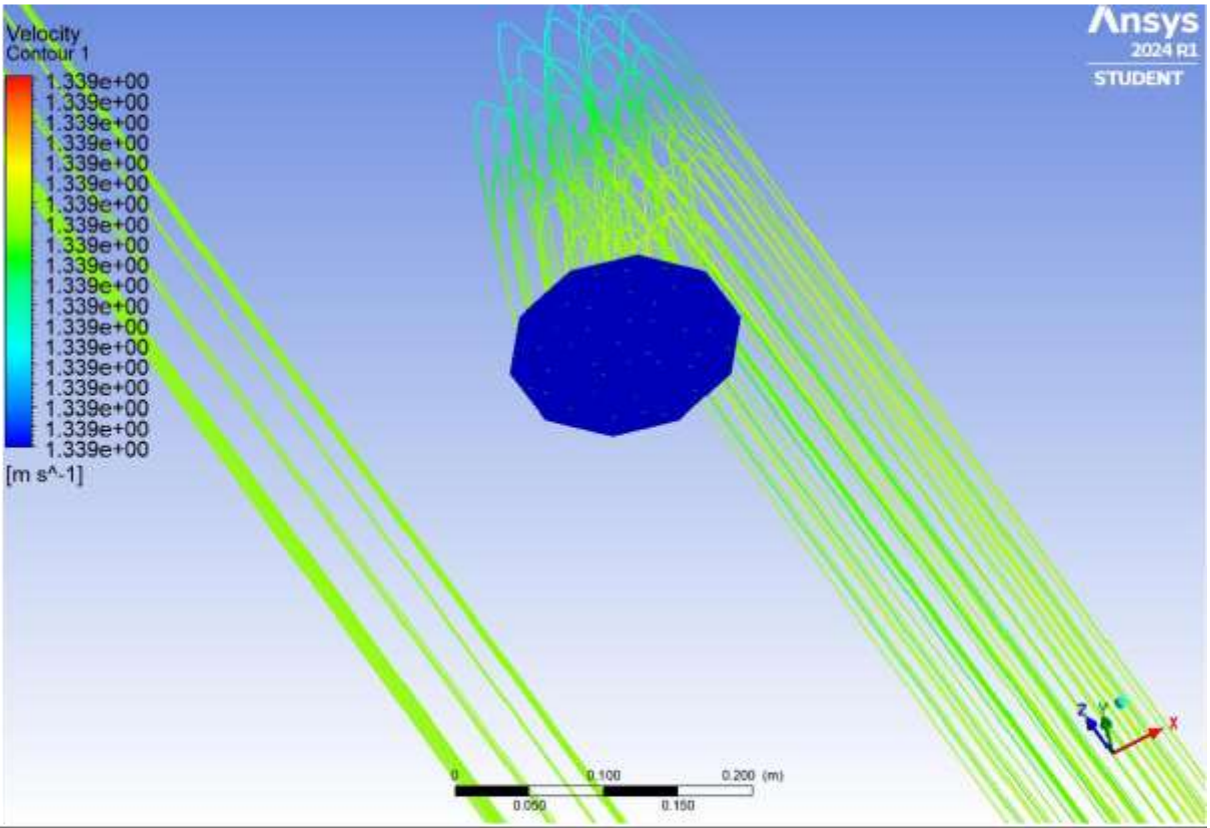

Figure 15:Velocity at the inlet of the pipe assembly

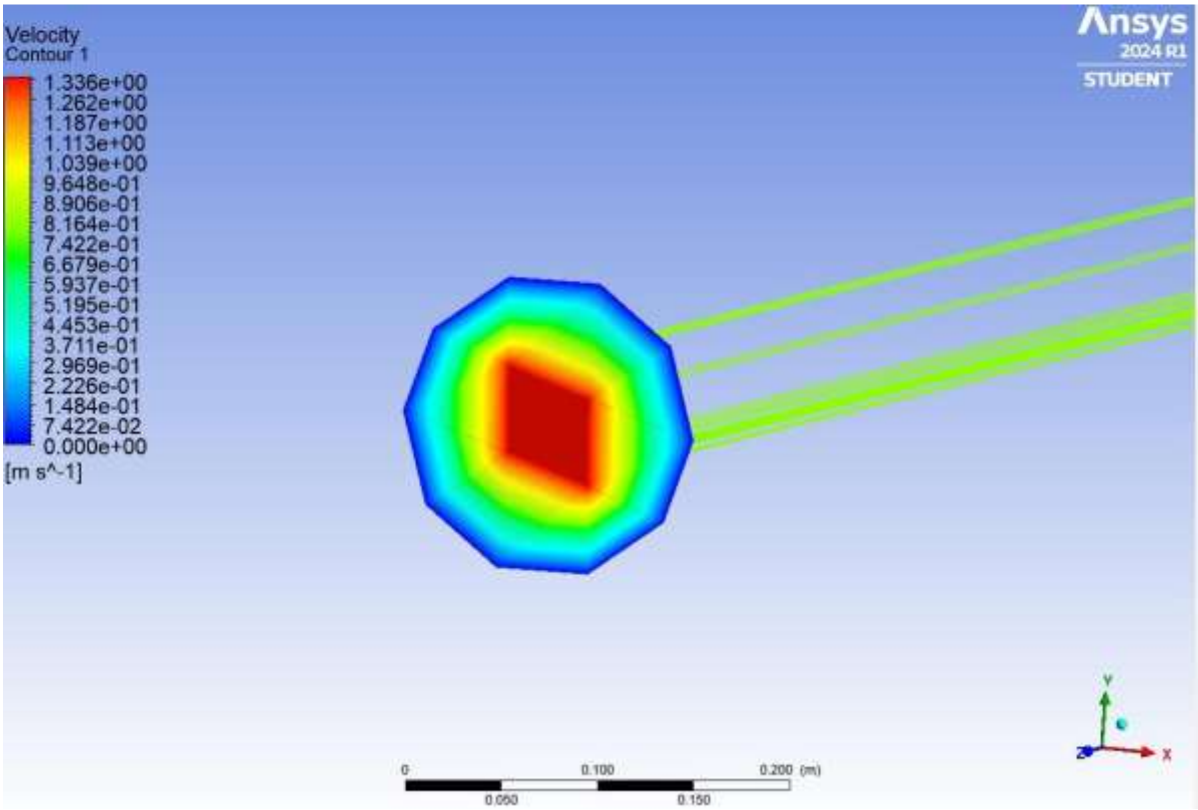

Figure 16: Velocity at the outlet of the pipe assembly

In figure 14 velocity gradually changes throughout the pipe due to change in diameter and heights in different sections, in the inlet section the velocity will be 1.338 m/sec, as the fluid enters from the inlet section to the SECTION 1 of the pipe assembly the velocity of the water will decrease to 1.2 m/sec, from SECTION 1 to SECTION 2 the velocity of the liquid will varies from 1.2 m/sec to 0.4 m/sec as the length of the pipe increases, from SECTION 2 to SECTION 3 the velocity of the water will varies from 1.2 m/sec to 0.7 m/sec as the length of the pipe increases, from SECTION 3 to SECTION 4 the velocity of the water will varies from 0.7 m/sec to 0.5m/sec as the length of the pipe increases, after the SECTION 4 water goes through SECTION 5, SECTION 6,SECTION 7, SECTION 8, SECTION 9 which contains the air handling units with smaller diameter i.e. 20 mm than the main pipe diameter 40 mm in these sections the velocity shows 1.1 m/sec and 1.4 m/sec, from the air handling units the water flows through the SECTION 10, SECTION 11, SECTION 12respectively and the velocity in this sections will be 0.3 m/sec,0.8 m/sec,0.3 m/sec respectively, then the water flows to the outlet section with velocity 1.1 m/s4.3.3 Temperature distribution analysis: Temperature distribution analysis in a CFD study of water-cooled chiller pipesprovides crucial insights into the thermal performance and heat transfer efficiency of the system. To investigate the heat transfer processes within the chiller, focusing on the efficiency of heat exchange between the building's cooling load and the water circulating through the chiller and to employ CFD simulations to analyze temperature distributions, heat fluxes, and thermal gradients across the chiller components. By visualizing temperature gradients along the length of the pipes, engineers can observe how the temperature of the water changes as it moves through the system. This visualization helps in identifying regions where the temperature remains higher or lower than expected, indicating potential inefficiencies in the cooling process. For instance, a steep temperature gradient might suggest effective heat exchange, whereas a shallow gradient could indicate poor heat transfer, possibly due to fouling or inadequate flow rates. Understanding these temperature variations is vital for ensuringthat the chiller operates within its optimal thermal range, preventing overheating and maintaining consistent cooling performance. Additionally, the analysis provides detailed insights into the effectiveness of heat transfer between the water and the pipe walls. This includes evaluating the local heat transfer coefficients at various points, which reflect how efficiently heat is being conducted away from the water. Areas where the heat transfer is less efficient can be pinpointed, allowing for targeted interventions such as improving the surface roughness of the pipes, adjusting the coolant flow rate, or even altering the pipe material to enhance thermal conductivity. These insights help in optimizing the design and operation of the chiller system to maximize its cooling capacity and energy efficiency. Moreover, engineers can anticipate possible trouble locations, such as hotspots that could result in thermal stress or component failure, by detecting regions where heat transport is less than ideal. All things considered, the temperature distribution analysis directs real-world modifications to produce more dependable and effective cooling performance in addition to improving knowledge of the thermal dynamics within the chiller system.

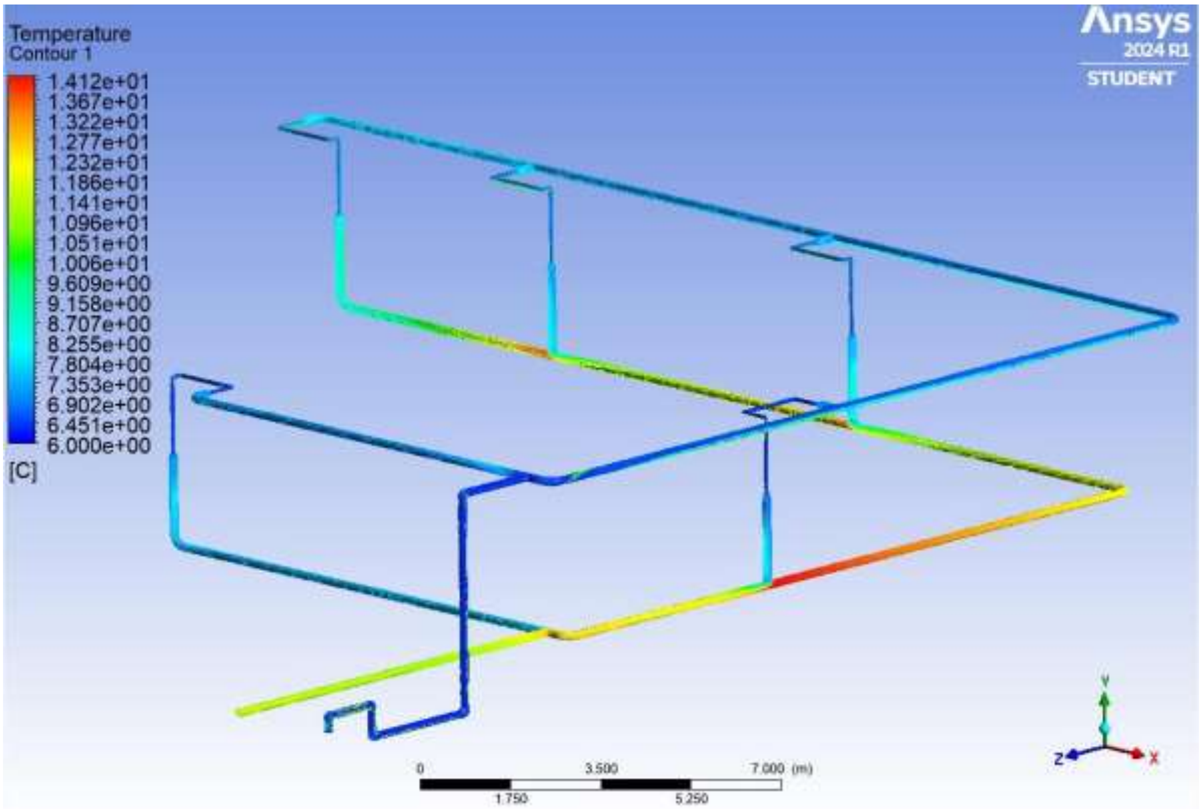

Figure 17: Temperature contour of the pipe assembly

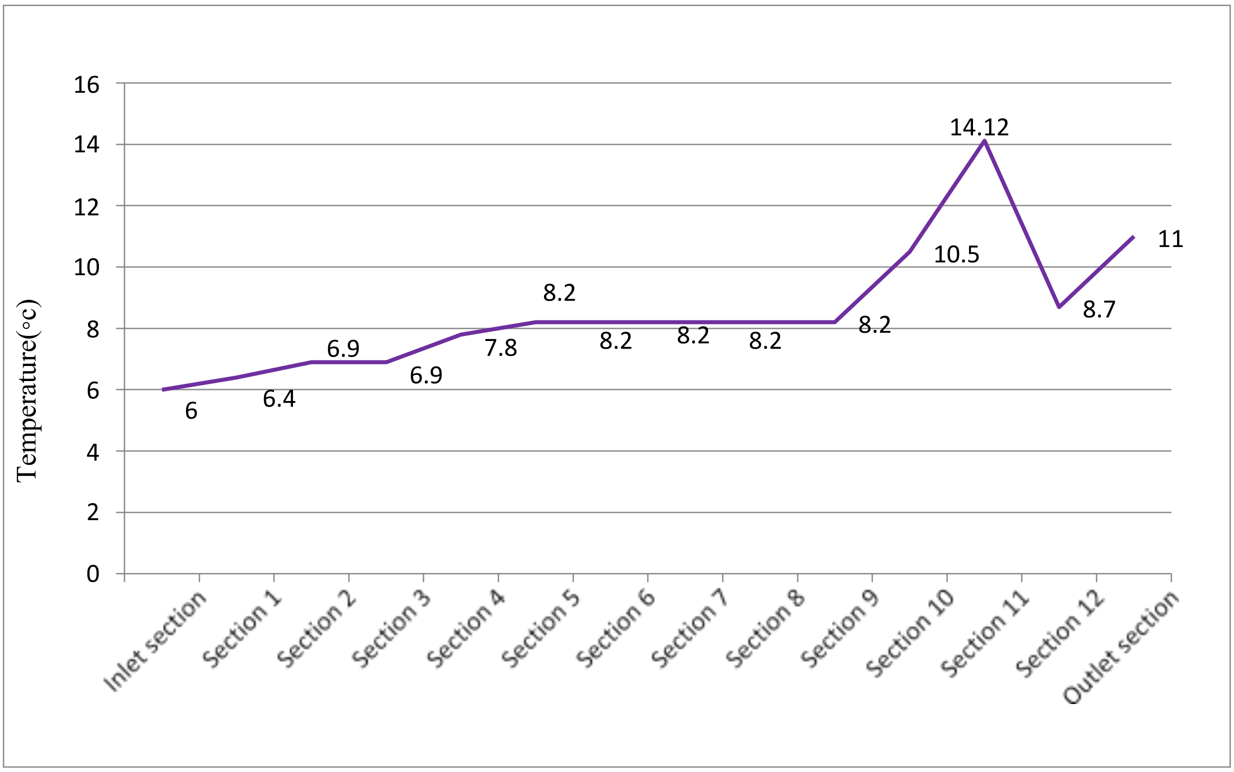

Figure 18:Variation of temperature throughout the pipe in different sections

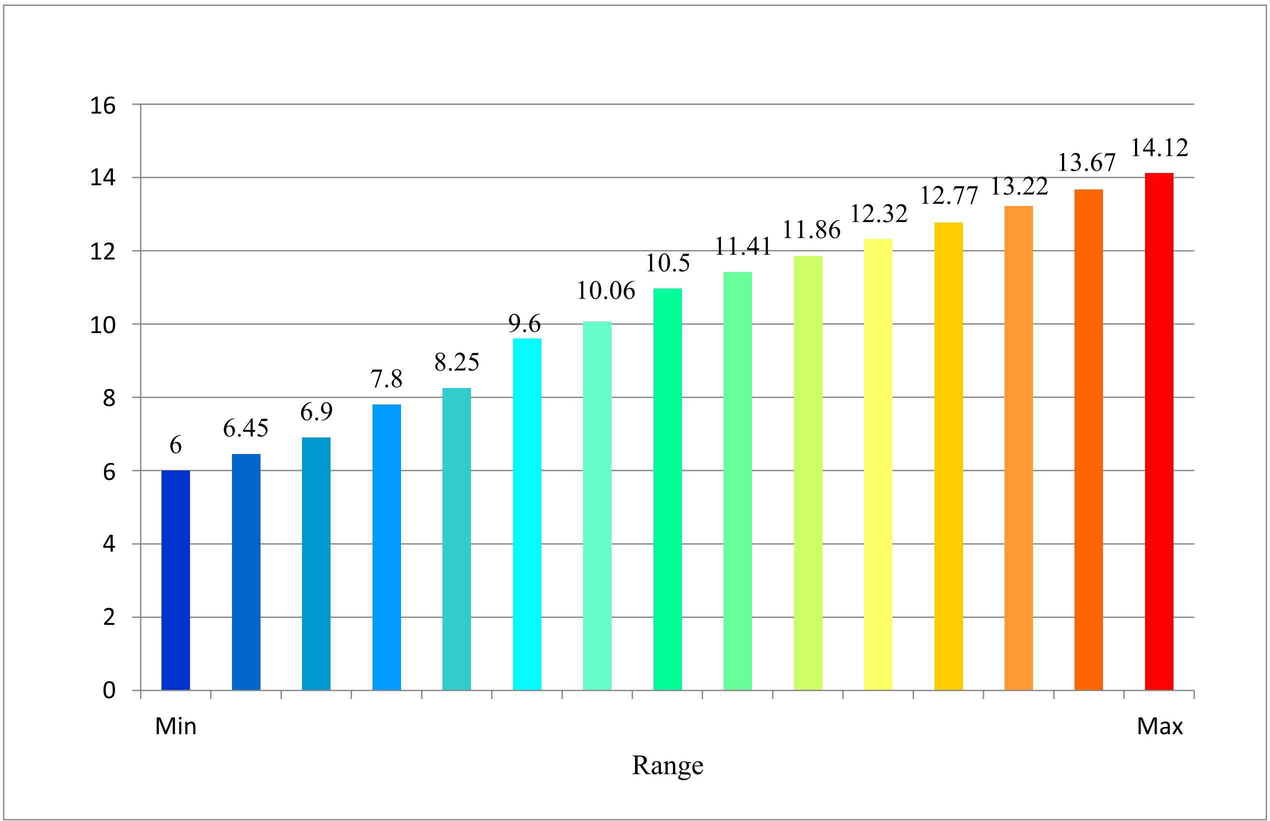

Figure 28: Variation of temperatures throughout the pipe in different sections

In the figure 28 from the temperature contour that occurs in ansys we can explain that the temperature at the inlet of the pipe will be 6◦c, as it flows from the inlet section to the SECTION 1 it gain some heat from the surroundings and attain the temperature as 6.4◦c, from SECTION 1 to SECTION 2 the temperature of the water will be 6.9◦c, now the water flows from SECTION 1 to SECTION 3 and the temperature will reach up to 6.9◦c, water now flows from SECTION 3 to SECTION 4 the temperature will vary from 6.9◦cto 7.8◦c, after this water will flow through SECTION 5, SECTION 6, SECTION 7, SECTION 8 and SECTION 9 where there will be 5 air handing unit where the water extract the heat from the from the room and where the temperature is 8.2◦c, after gaining the temperature the water flows to the SECTION 10 with temperature 10.5◦c and to the SECTION 11 with highest temperature 14.12◦c and in SECTION 11 the temperature will be 8.7◦c, after mixing ,all the water goes through the outlet section which has the temperature as 11◦c.4.4

Conclusion:

The conclusion summarizes the key findings and progress of the work, highlighting its success in addressing the objectives. determination of Water Flow Rate: The calculation of the needed water flow rate for the water-cooled chiller system is a critical milestone in this project since it directly effects the system's ability to efficiently disperse heat absorbed from the building. Water is essential in transporting away heat created during the cooling process in a water-cooled chiller system. The exact computation of the water flow rateis critical to the efficiency of the chiller system.

The water flow rate is the amount of water that must be circulated through the chiller system per unit of time. It is determined by elements such as the building's cooling demand, the efficiency of the chiller, and the specific heat capacity of water. The purpose is to guarantee that enough water flows through the chiller to absorb and transport the heat created by the chilling process.

The exact estimation of the water flow rate is crucial for the chiller system's optimal performance. If the flow rate is too low, the chiller may struggle to efficiently remove heat, resulting in decreased chilling capacity and associated inefficiencies. Onthe other side, excessive flow rate may result in wasteful energy consumption and operational costs.

This project ensures that the water-cooled chiller system functions at top efficiency by effectively identifying the needed water flow rate. This not only improves energy economy by maximizing the chiller's cooling capability, but it also aids in the maintenance of a constant and controlled indoor temperature within the office building. The accuracy with which the water flow rate is determined improves the cooling system's overall performance by aligning it with the calculated cooling load and supporting a sustainable and energy-efficient approach to temperature regulation in the office area.

5.3 CFD simulation: The CFD analysis of the pipes in a water-cooled chiller has provided a comprehensive understanding of the fluid dynamics, thermal characteristics, and performance inefficiencies within the system. The detailedmapping of pressure distribution along the pipes revealed crucial insights into thevariations in pressure, highlighting areas of significant pressure losses due to friction,bends, and fittings. This understanding is critical for ensuring proper pump sizing andoptimizing energy efficiency, as it allows for precise quantification of both frictionaland minor losses. Such insights enable targeted design improvements that canminimize unnecessary pressure drops, thereby enhancing overall system performance.

The CFD simulations' flow distribution and velocity profiles showed how crucial uniform flow is to attaining effective heat transfer. The research identified particular spots where flow disruptions occur, potentially resulting in thermal inefficiencies and system failures. These locations include areas of turbulence, recirculation, and low flow regions. These results highlight the necessity of flowmanagement techniques and improved pipe shapes in order to prevent hotspots and ensure uniform flow distribution.

The analysis of temperature distribution has enhanced our comprehension of the thermal dynamics in the chiller system. Through the visualization of temperature gradients along the pipes, the study identified areas where heat transfer was either very effective or not very efficient. As a result, possible hotspots and locations for better cooling performance might be found. Understanding the local heat transfer coefficients gave rise to a foundation for improving the system's thermal conductivity through surface treatment, material selection, or operational modifications.

In summary, the CFD study of a water-cooled chiller's pipes has produced insightful information that can direct the development, enhancement, and use of more dependable and efficient cooling systems. The total efficiency and performance of the chiller can be greatly enhanced by resolving the pressure losses, flow irregularities, and thermal inefficiencies that have been observed. These results not only advanceour knowledge of the dynamics of chiller systems, but they also lay a strong basis for further investigation and improvement of the efficiency of water-cooled chillers. The study's recommendations and optimizations have the potential to save operating costs,increase system lifespan, and support more environmentally friendly cooling systems.

REFERENCES

1. Lei, Z., & Zaheeruddin, M. (2005). Dynamic simulation and analysis of a water chiller refrigeration system. Applied Thermal Engineering, 25(14-15), 2258-2271.
2. Olivieri, S. J., Henze, G. P., Corbin, C. D., & Brandemuehl, M. J. (2014). Evaluation of commercial building demand response potential using optimal short-term curtailment of heating, ventilation, and air-conditioning loads. Journal of Building Performance Simulation, 7(2), 100-118.
3. Mumtaz, H., & Hammdi, S. (2024). Numerical simulation of integrating an air conditioner with an evaporative air cooler. Heritage and Sustainable Development, 6(1), 43-66.
4. Hassanzadeh, R., Shams, M., & Darvishyadegari, M. (2019). An environmentally friendlyidea for cooling of water in water-cooled chillers in urban areas. International Journal of Advanced Research in Physical Science, 6(1), 7-18.
5. Hashim, H. M., Sokolova, E., Derevianko, O., & Solovev, D. B. (2018, December). Cooling load calculations. In IOP Conference Series: Materials Science and Engineering (Vol. 463, No. 3, p. 032030). IOP Publishing.
6. Suamir, I. N., Ardita, I. N., & Rasta, I. M. (2018, October). Effects of cooling tower performance to water cooled chiller energy use: a case study toward energy conservation of office building. In 2018 International Conference on Applied Science and Technology (iCAST) (pp. 712-717). IEEE.
7. Sharma, S. K. Thermodynamic Investigations of water cooler Chiller Plant of an air- conditioning System. International Journal of Engineering and Applied Sciences, 4(4), 257478.
8. Asfand, F., Stiriba, Y., & Bourouis, M. (2015). CFD simulation to investigate heat and mass transfer processes in a membrane-based absorber for water-LiBr absorption cooling systems. Energy, 91, 517-530.
9. Zhu, N., Hu, P., Lei, Y., Jiang, Z., & Lei, F. (2015). Numerical study on ground source heat pump integrated with phase change material cooling storage system in office building. Applied Thermal Engineering, 87, 615-623.Gang, W., Wang, S., Shan, K., & Gao, D. (2015). Impacts of cooling load calculation uncertainties on the design optimization of building cooling systems. Energy and Buildings, 94, 1-9.
10. Chang, Y. C., Chen, C. Y., Lu, J. T., Lee, J. K., Jan, T. S., & Chen, C. L. (2013). Verification of chiller performance promotion and energy saving.
11. Hua, T., Yitai, M., Minxia, L., Chuntao, L., & Li, Z. (2010). The status and development trend of the water chiller energy efficiency standard in China. Energy Policy, 38(11), 7497-7503.
12. Fouda, A., & Melikyan, Z. (2010). Assessment of a modified method for determining the cooling load of residential buildings. Energy, 35(12), 4726-4730.
13. Tashtoush, B., Molhim, M., & Al-Rousan, M. (2005). Dynamic model of an HVAC system for control analysis. Energy, 30(10), 1729-1745.
14. Chaerun Nisa, E., & Kuan, Y. D. (2021). Comparative assessment to predict and forecast water-cooled chiller power consumption using machine learning and deep learning algorithms. Sustainability, 13(2), 744.
15. Zinet, M., Rulliere, R., & Haberschill, P. (2012). A numerical model for the dynamic simulation of a recirculation single-effect absorption chiller. Energy Conversion and Management, 62, 51-63.
16. Trčka, M., & Hensen, J. L. (2010). Overview of HVAC system simulation. Automation in construction, 19(2), 93-99.
17. Yu, F. W., & Chan, K. T. (2008). Optimization of water-cooled chiller system with load-based speed

control. Applied Energy, 85(10), 931-950.
18. Koury, R. N. N., Machado, L., & Ismail, K. A. R. (2001). Numerical simulation of a variable speed refrigeration system. International journal of refrigeration, 24(2), 192-200.
19. Lu, L., Cai, W., Xie, L., Li, S., & Soh, Y. C. (2005). HVAC system optimization—in-building section. Energy and Buildings, 37(1), 11-22.
20. Beyene, A., Guven, H., Jawdat, Z., & Lowrey, P. (1994). Conventional chiller performances simulation and field data. International Journal of Energy Research, 18(3), 391-399.
21. Kohlenbach, P., & Ziegler, F. (2008). A dynamic simulation model for transient absorption chiller performance. Part I: The model. International journal of refrigeration, 31(2), 217-225.Kohlenbach, P., & Ziegler, F. (2008). A dynamic simulation model for transient absorption chiller performance. Part II: Numerical results and experimentalverification. International journal of refrigeration, 31(2), 226-233.
22. Crowther, H., & Furlong, J. (2004). Optimizing chillers & towers. ASHRAE Journal, 46(7), 34.
23. Satish, G., Kumar, K. A., Prasad, V. V., & Pasha, S. M. (2013). Comparison of flow analysis of a sudden and gradual change of pipe diameter using fluent software. IJRET, 2,41-5.
24. Martins, N. M., Carrico, N. J., Ramos, H. M., & Covas, D. I. (2014). Velocity- distribution in pressurized pipe flow using CFD: Accuracy and mesh analysis. Computers & Fluids, 105, 218-230.
25. Ben-Nakhi, A. E., & Mahmoud, M. A. (2004). Cooling load prediction for buildings using general regression neural networks. Energy Conversion and Management, 45(13- 14), 2127-2141.
26. Kaya, A. (1991). Improving efficiency in existing chillers with optimization technology. ASHRAE Journal (American Society of Heating, Refrigerating and Air- Conditioning Engineers);(United States), 33(10).
27. Yu, F. W., & Chan, K. T. (2010). Economic benefits of optimal control for water- cooled chiller systems serving hotels in a subtropical climate. Energy and Buildings, 42(2), 203-209.
28. Kim, B., & Park, J. (2007). Dynamic simulation of a single-effect ammonia–water absorption chiller. International Journal of Refrigeration, 30(3), 535-545