

COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS OF COLD STORAGE**¹Chandrashekhhar Meshram, ²Onkar Borde, ³Kartik Pandit, ⁴Kunal Marathe, ⁵Prof. Ajay Waychal**

U. G Student Mechanical Department, JSPM's Imperial College of Engineering & Research, Pune, India^{1,2,3,4}, Professor Mechanical Department, JSPM's Imperial College of Engineering & Research, Pune, India⁵
shekharsmeshram@gmail.com¹, onkarborde63@gmail.com², kartikpp2000@gmail.com³,
ksmarathe@gmail.com⁴, ajay.g.waychal@gmail.com⁵

ABSTRACT

In this project we deal with the numerical analysis of 3D model of a cold storage which is developed to study the effects of design parameters on flow and temperature fields within the cold storage. The variation in number of inlets, position of inlets and stacking methodology affect flow and temperature fields within the cold storage. Results of CFD simulation indicated the optimum position of the inlets, number of inlets and stacking methodology required to have uniform flow and temperature field within the cold storage and try to improve the efficiency of cold storage. The objective of this project is to design and improve the efficiency of cold storage for getting more thermal comfort, air circulation and reducing contamination in cold storage application. A tetrahedral mesh is going to create by using ANSYS 21.0 and calculation finished when accessing a solution. Turbulence is modelled using the (k-epsilon). The present study reports on velocity and temperature monitoring in cold storage.

Keywords: *Computational fluid Dynamics, Ansys, cold storage, preservation of apple.*

INTRODUCTION

A cold storage is a place where the various items such as vegetables fruits, medicines etc. are stored to protect them from getting spoiled and to prolong their preservation period. This is done by storing the products at their preservation temperature and humidity etc using proper cooling system. Many cooling systems are used in cold storages; one of them is Evaporator coil located at the top of one of the Wall of cold storage. We have analysed this cooling system on CFD.

Computational fluid dynamics (CFD) uses powerful computer and applied mathematics to model fluid Flow problems. In the recent years CFD has been applied in the food processing industry. Numerical modelling of airflow and temperature distribution in a cold store was performed using the Computational Fluid Dynamics (CFD). The aspects which were investigated include the influence of wind velocity. A mono-scale three-dimensional Computational Fluid Dynamic model was developed for estimating of airflow, heat and mass transfer. A computational Fluid Dynamics model was developed to estimate distribution of temperature and relative humidity.

Quality and safety of perishable products such as apple during storage is highly dependent on temperature and temperature fluctuations. The present study reports on temperature monitoring in cold storage and containers which are used for storage. Considerable temperature fluctuations were observed, especially in some of the cold stores. The aspects which were investigated include the influence of wind velocity inside the building.

LITERATURE REVIEW**An Analysis of Cold Store by CFD Simulation: -**

Dr. B. Sadashive Gowda Principal, Vidyavardhaka College of Engineering, Mysuru, India has done this study where the numerical analysis of 3d model of a cold store of dimensions 4.5 m (L) x 3.3 m (W) x 2.5 m (H), which is developed to study the effects of design parameters on flow and temperature fields within the cold store. The variation in number of inlets, position of inlets and stacking methodology affect flow and temperature fields within the cold store. Results of CFD simulation indicated the optimum position of the inlets, number of inlets and

stacking methodology required to have uniform flow and temperature field within the cold store. It also proved that CFD is a convenient tool for designing and optimizing the flow field in the cold store. The problem is solved using ANSYS CFX and the code is validated with the results of a research paper.

In the present study the design parameters affecting the flow and temperature fields are studied. The design parameters include the position of the inlet, number of the inlets and outlets and the stacking methodology. It aims to have uniform air flow and temperature distribution within the cold store and thus prevent hot spots on the surface of the produce. The following designs are considered for analysis-

- A) To change the position of the inlet and outlet in case of single inlet type of cold stores.
- B) To have different configurations of multiple air inlets and outlets instead of single inlet and outlet and changing the methodology.

CFD Modelling of Two Different Cold Stores Ambient Factors

Serap Akdemira, Serhat Ozturkc, Firat Oguz Edisb, Erdinc Bald a Asssit.Prof.Dr., Namik Kemal University, Technical Sciences Vocational School, Air-Conditioning and Refrigeration Program, 59030, Tekirdag, Turkey. Objective of the research was to determine ambient temperature and relative humidity distributions of two different cold stores which have two different cooling systems. One of the cold Store which is called as Cold store-I, has classical cooling system such as compressor, condenser and evaporator. Second called Cold store-II, has air conditioning system for cooling, cold air ventilation and aspiration systems, and humidification system. Computational fluid dynamics was used for modelling of distribution of temperature and relative humidity of cold store walls. Storage temperature and relative humidity were assumed 2 °C and 90%, respectively.

Boundary conditions were set as; Inlet-Surface of fluid inlet, Outlet-Surface of fluid outlet, and walls-solid, proof against flow of fluid. A tetrahedral mesh was created by using ANSYS 14.0 and calculation finished when accessing a solution. Turbulence was modelled using the(k-epsilon). Spatial distribution in two cold stores for two different cooling systems were modelled and evaluated in this research. Data determined from CFD models were compared for both cold stores. Cold store-II was better than Cold store-I because it has air distribution holes located on ceiling.

Cold store-I: - The cold store which is called as Cold store-I, has classical cooling system such as compressor, condenser and evaporator. Sizes of cold store-I were 4.60X 4.35X 3.41 m (in length, width and height) and its volume was 68.3 m³. Type and power of Compressor were hermetic and 7BG, respectively. Condenser has axial fan and its capacity was 15 kW.

Cold store-II: - Sizes of the cold store were 4 x 5 x 3 m(in length, width and height) Cold store volume was 60 m³. Cooling capacity was 15KW and cold air ventilated into cold store by inlet air channels. Consequently, homogeneity distribution of cold air establishes in cold store Working temperature and relative humidity varied between 0°C /+30 °C and 55-95 %, respectively. Tolerance of relative humidity was ± % 5 and ± 0,5 °C. The system consists of water-cooling unit, air conditioning unit and control unit.

Methods Modelling of spatial distribution of temperature and relative humidity in these two cold stores were realized by using computational fluid dynamic (CFD). Software was Ansys Fluent 14.0.

Computational analysis includes stages of geometrical modelling, creating numerical mesh, analysis and evaluating of results. This process requires powerful computer system. In this research, this methodology was used to analyse the cold stores. For this purpose, boundary condition of the cold storages was determined. Boundary conditions are; Inlet; Surface of fluid inlet, Outlet; Surface of fluid outlet, Walls; Solid, proof against flow of fluid

A tetrahedral mesh was created and calculation finished when accessing a solution. Turbulence was modelled using the (k-epsilon). Heat transition from boundaries and lightening system were taken into account for modelling. Base of cold stores were accepted as isolated surface. As a boundary condition, constant heat transition was used for walls and ceiling. Data were determined from 18 different points and evaluated by using descriptive statistics for each cold storage.

METHODOLOGY: -

1. Creating a virtual model.
2. The flow region or calculation domain is divided into a large number of finite volumes or sales.
3. Partial differential equations are discretized using a wide range of techniques: finite difference, finite volume or finite element.
4. Algebraic equations gathered into matrices which are solved by an iterative procedure.
5. Numerical solution gives the value of the depended variables at discrete location.
6. Chemical reaction, multiphases flow, mixing, phase change, mechanical movement are studied.
7. Obtaining plots, charts of the optimized results.

Virtual Geometry Creation by Using Ansys 2022 R1: - Three-dimensional model description

1. A Closed Room
2. A Fan Coil Unit
3. A Door
4. 33 Racks
5. Inlet
6. Outlet

Several Terms Are Used to Describe Cold Storage: -

1. Dimensions of cold storage => length=10m, width=5m and height=4m
2. Capacity of cold storage = 20,000 Kg (20 tons)
3. Area of walls and roof = 210 m²
4. Area of floor = 50 m²
5. Volume of plant = 200 m³
6. Ambient air = 30°C at 50% RH
7. Internal air = 1°C at 95% RH
8. Insulation = polyurethane 80mm thick
9. U value =0.28 w/m²k Floor temp. =5°C

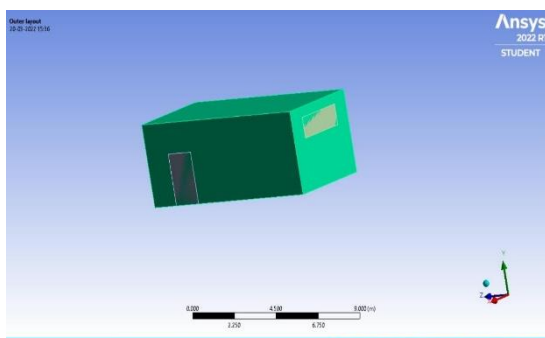


Figure 1: Outer 3D model of cold storage

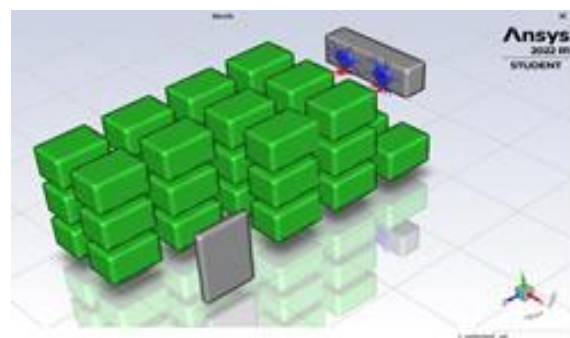


Figure 2: Internal 3D model of cold storage

Mesh Generation

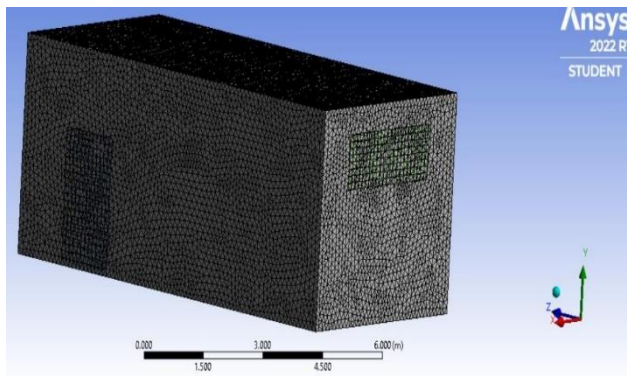


Figure 3: Meshing

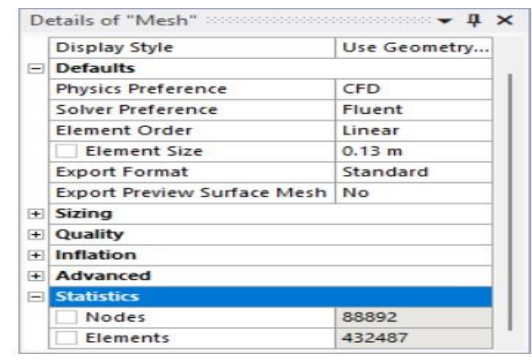


Figure 4: Details of Mesh

SIMULATION IN FLUENT SETUP

By selecting the setup command after mesh, we will have to give the boundary conditions which will help to give our specific conditions for the give simulation. The mesh file created in meshing is updated and then this mesh is used for the setup in fluent. It allows us to give pressure, velocity, temperature and many more such conditions for the specific product or geometry. It also allows us to select our display options, processing options.

Post Processing

For the post processing 3D Geometry is converted into 2D plane using slicing and symmetry tools and mesh is generated of this 2D plane.

Boundary Conditions: -

Velocity at inlet = 3 m/s

Inlet temperature = 280 K

Scaled Residuals: -

Here in scaled Residuals, it is observed that the energy goes on decreasing, k-omega scales are higher than the Velocities in x and y direction. The iterations till 500 are done.

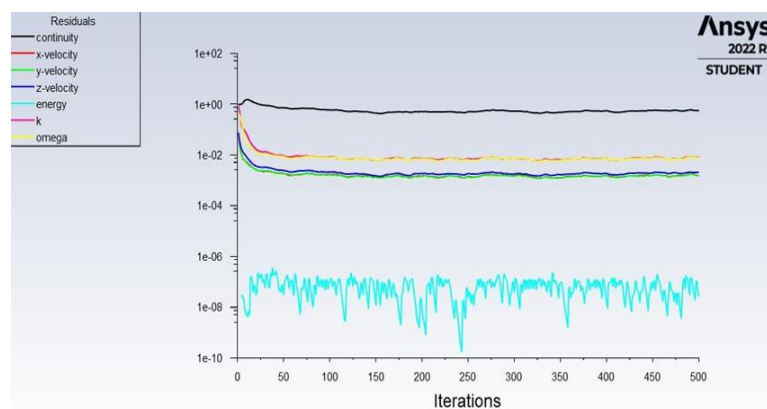


Figure.5: Scaled Residual

Velocity: -

The velocity at the inlet is less than outlet which is 3 m/s and maximum at the outlet which is 4.388 m/s the increase in velocity at outlet.

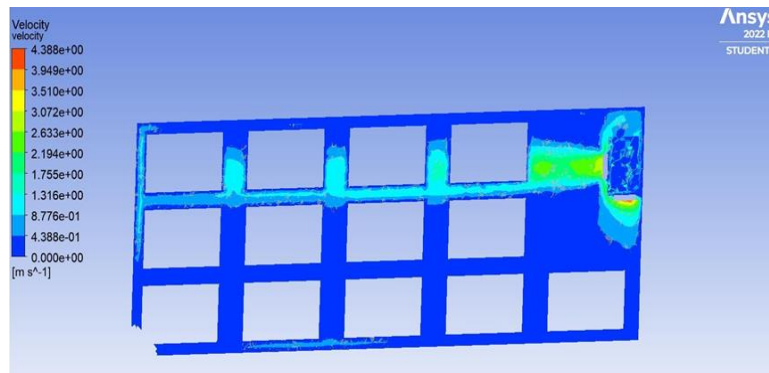


Figure 6: Velocity

Pressure: -

Pressure at the inlet is 26.14 Pa and Minimum at the outlet which is 2.854 Pa. here in Front portion of the fan coil unit pressure is more which is 37.78 Pa and in back side of it is 26.14 Pa.

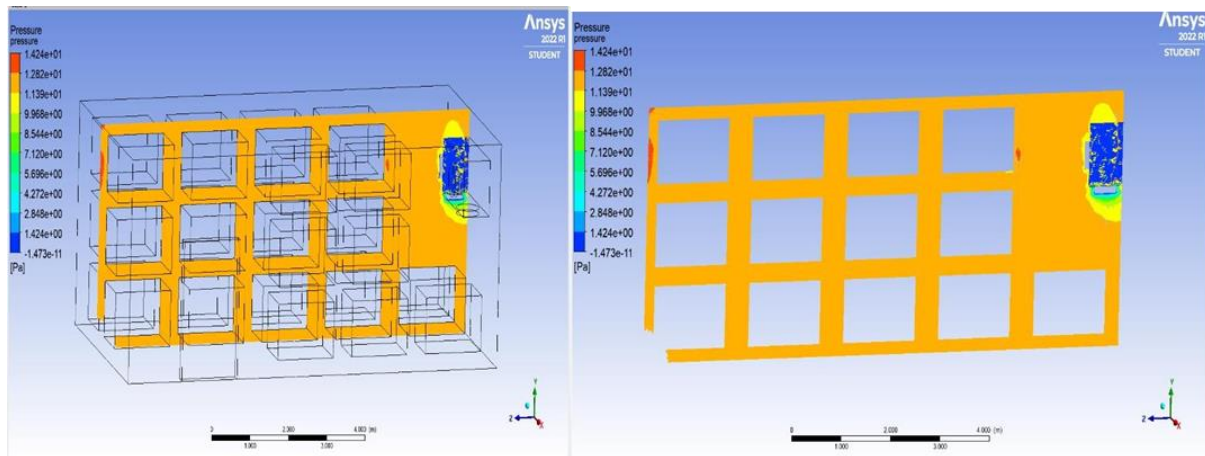


Figure 7: Pressure

Temperature Contour: -

Temperature in the cold room is maintained constant which is 280 K and some temperature fluctuation is there at the outlet for very small region it varies from 300 K – 274.1 K.

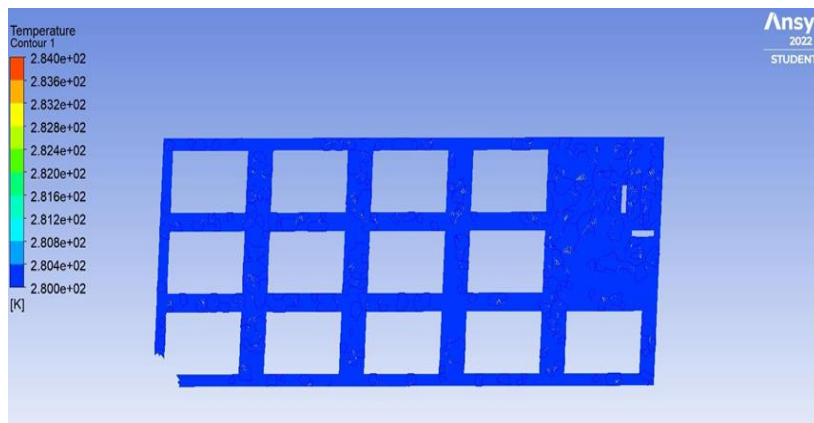


Figure 8: Temperature Contour

Velocity Streamline: -

From the velocity streamline it is seen that how the flow of velocity of air varies for different positions in cold storage. The streamline velocity is varying between 0 m/s – 4.388 m/s around the racks inside the cold storage.

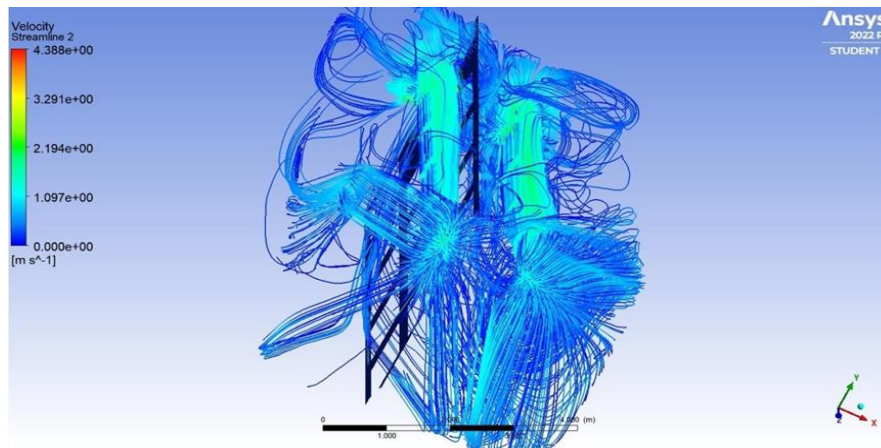


Figure 9: Velocity Streamline

CONCLUSION

This study presents idea about the flow of air velocity, temperature and pressure in cold Storage. The main objective is to determine the flow of velocity, temperature and pressure inside the cold Storage room using k-epsilon model. From the study it is concluded that in the storage room pressure and velocity at different positions are different. Velocity near the fan coil unit is maximum and pressure near the fan coil unit is less than that of the other area where the pressure is more but the velocity is less. Required Temperature which is to be maintained in the storage room is remains constant for complete cold storage room. The variation in velocities is from 0.4388 m/s – 4.388 m/s. variation in pressure is from 2.848 Pa – 14.24 Pa.

REFERENCES

- [1] DrB.Sadashive Gowda, “An Analysis of Cold Store by CFD Simulation”, principal, Vidyavardhaka college of engineering, Mysuru, India, May 2015.
- [2] Prashant Jaiswal, Rohit Soni, Ravi Vishwakarma, “Flow Analysis of Two Different Evaporator Arrangements in a Cold Storage Using CFD”, research scholar, professor, Shree institute of science and technology, RGPVBhopal, MP, (India), 2013.
- [3] Bjorn Margeirsson and Sigurjon Arason, “Temperature Monitoring and CFD Modelling of a Cold Storage”, Matis ohf. / University of Iceland, Borgartun 21, 105 Reykjavik, Iceland, June 2008.
- [4] Serap Akdemira, Serhat Ozturk, Firat Oguz Edis, Erdinc Bal, “CFD Modelling of Two Different Cold Stores Ambient Factors”, Asssit. Prof.Dr. Namik Kemal University, Technical Sciences Vocational School, Air-Conditioning and Refrigeration Programme, 59030, Tekirdag, Turkey, (2013).
- [5] Cinzia Buratti, Domenico Palladino, Elissa Moretti, “Prediction of Indoor Conditions and Thermal Comfort Using CFD Simulations: A Case Study Based on Experimental Data”, Department of Engineering, University of Perugia, Via G. Duranti 93, 06125 Perugia (PG), Italy, September 2017, Lecce, Italy.
- [6] Sumesh Pillai, Saurabh Bhand, Vikas Shinde, Siddharth Pawar and Pravin Katare “A Review on CFD Analysis in Air-Conditioning System”, Mechanical Engineering Department, Marathwada Mitra

Mandal's Institute of Technology, SPPU, Pune-47, India, Accepted 01 Oct 2016, Available Online 05 Oct 2016, Special Issue-6 (oct 2016).

- [7] Razvan Mahu, Florin Popescu, Ion V. Ion, "CFD Modelling Approach for HVAC Systems Analysis", Dunarea de Jos university of Galați, 47 Domneasca St. 800008, Galați, Romania, January 2012.

