IJEAST

IJEAST

INTERNATIONAL JOURNAL OF ENGINEERING APPLIED SCIENCE AND TECHNOLOGY

VOLUME : 9 ISSUE : 10 Print / Issue Publication Date: 21-Apr-2025

ISSN : 2455-2143

DOI : 10.33564/IJEAST.2025.v09i10.003

Indexed In

Google SCHOLAR OPEN ACCESS doi Crossref

WWW.IJEAST.COM

editor@ijeast.com

International Journal of Engineering Applied Sciences and Technology, 2025
Vol. 9, Issue 10, ISSN No. 2455-2143, Pages 30-41
Published Online February 2025 in IJEAST (http://www.ijeast.com)

NUMERICAL SIMULATION OF EXHAUST POSITION FOR OPTIMIZED VENTILATION IN A TEXTILE MANUFACTURING PLANT: A CFD STUDY

Abinash Verma, Sanjeev Gupta
Mechanical Engineering Department
GCET Jammu, J & K(UT), India

Abstract—**Textile industry is one of the highest employment generators in India, however most of the industries involved in this sector are MSME with limited production rates. These industries commonly utilise the traditional methods of production which are labour intensive and slow and as such there is limited scope for growth and profit. Due these factors, the worker comfort and general well-being is often an afterthought which is itself one of the reasons for the less production rate of the industries. It is an established fact that the production rates are directly related to the working environment of the workers i.e., better working environment results in better production rate. So, to ensure better production rates as well as worker well-being it is important to consider installation of a heating, ventilation, and air conditioning system. This is especially true for a sub-tropical country like India where humidity and temperature variations impact not only the workers' productivity but also the quality of the textile industry produce. Materials like cotton, wool, silk etc, are affected by the humidity as well as the temperature variation and as such it is imperative to ensure a balanced and optimised environment in the textile industry. In this study a basic model based on a small textile factory with natural ventilation is prepared and then modified to incorporate mechanical ventilation system with only one inlet and outlet each to keep the cost low. By modifying the position of the outlet with respect to the loom different ventilation systems are prepared and then CFD simulations are run to identify the optimised ventilation system. The goal is to establish an optimised and cost-effective ventilation system for a small textile industry.**

Keywords—**CFD analysis, exhaust position, productivity, textile industry, ventilation**

I. INTRODUCTION

Productivity in any industry is directly related to the working environment of the workers and this is true even in the textile industry. Having a comfortable working environment (temperature, humidity, lightening level, and sound level within acceptable limits) directly results in increased productivity and thereby better profits for industries. Environmental factors also play a crucial role in the textile manufacturing as well [**Error! Reference source not found.**]. Therefore, it is important to ensure that the working environment is controlled and optimised in a textile industry to ensure maximum efficiency and profits.The textile sector is the second largest employer after agriculture in India and of which most of the workers are engaged in the Micro, Small, and Medium Enterprise (MSME) sector. Due to financial constraints in most MSME textile industries the worker comfort is often an afterthought and not much focus is placed on it. These industries are focused on cheap labour and manual work rather than automation which limits the production rates. In addition, these industries comprise of non-integrated units capable of performing only one or two operations like spinning, weaving, or finishing etc. [**Error! Reference source not found.**]. To increase the production rates automation and worker comfort measures need to be introduced. One way to ensure worker comfort is via installing a ventilation system. Broadly ventilation systems are of two types natural (traditional) ventilation and forced (mechanical) ventilation. Natural ventilation typically involves three types of effects- wind, temperature difference, and buoyancy. Natural ventilation offers reduced energy use and cost with the downside being its ineffectiveness to deal with humidity [**Error! Reference source not found.**].

Forced ventilation systems utilize the mechanical components (fans) to move the air and thereby provide ventilation. Two most common used forced ventilation systems in industry are extract ventilation and spot cooling [**Error! Reference source not found.**]. Extract ventilation systems are relatively simple and inexpensive to install [**Error! Reference source not found.**]. They utilize the natural or mechanical supply of air to ensure thermal comfort with the drawback being potential supply of pollution/dust with the air [**Error! Reference source not found.**]. Spot cooling ventilation focus on delivering comfort to the workers in an energy efficient manner by only cooling the areas where the workers operate and not cooling the whole area [**Error! Reference source not found.**]. In addition, compared to wall fan extract ventilation and roof fan extract ventilation system spot cooling system is more effective [**Error! Reference source not found.**]. To optimize the ventilation systems air flows needs to be studied. Air distribution is based on Navier-Stokes equation which can be solved for simple and ideal conditions but not for complex geometries. This resulted in the need to solve complex numerical equations which ultimately can be simplified by utilizing CFD to understand air distribution [**Error! Reference source not found.**]. To simulate air flow in a room turbulence models must be prepared, and solved. Broadly two types of models are used- Reynolds-Averaged Navier-Stokes (RANS) Models – EVM, NLEVM, DSM (RSTM or SOC) or Computation of fluctuating quantities – LES or DNS. Any CFD problem consists of four main components: geometry and mesh formation, setting-up a physical model, solving it and post-processing the computed data. Precise theory is available for geometry and mesh generation but not so for setting up the physical model for turbulence flows. Only simple models can be used for determining complex turbulence phenomenon and hence it requires a combination of least complex modelling equation and core of relevant physics [**Error! Reference source not found.**].

Room ventilation has three basic components [**Error! Reference source not found.**] ventilation rate — the amount of outside air that enters the room, as well as the quality of the outside air; airflow direction — the net airflow direction in a room, which generally should be from clean places to dirty places; air distribution or airflow pattern — the outside air should be delivered to each and every part of the room in an efficient manner and the airborne pollutants produced inside the room should also be removed in an efficient manner. Thermal comfort in a room depends not only on the temperature of the air and its velocity but also by turbulence intensity of air motion [**Error! Reference source not found.**]. In addition, the dimension and position of diffuser also influences the thermal comfort in a room [**Error! Reference source not found.**]. Any obstacle present in the direction air flow causes its deflection and as such has an impact on the distribution of air [**Error! Reference source not found.**]. All this is difficult to predict and solve if one performs physical model tests and hence the use of CFD provides an enhancement to the design process by giving more qualitative and quantitative information regarding air distribution and air flow [**Error! Reference source not found.**].Ventilation effectiveness is also a factor that can be evaluated by either measurement or simulation [**Error! Reference source not found.**]. Generally, the flow rate of ventilated air is be measured by measuring how quickly injected tracer gas decays in a room, or by measuring the velocity of air through ventilation openings or air ducts, as well as the flow area. Traditionally, the direction of airflow is visualized by smoke [**Error! Reference source not found.**]. Computational fluid dynamics [**Error! Reference source not found.**] and particle image velocimetry techniques [**Error! Reference source not found.**] can also be used to model the distribution of air in a room as well as check its performance.

In this paper, a small textile industry room with a mechanical ventilation system installed has been modelled and a CFD based analysis is performed using three different simulations as per the three different outlet positions of the ventilation system.

II. PROPOSED ALGORITHM

The figure 1 gives a flowchart of the steps involved in performing a CFD analysis of the ventilation system in a small textile factory.

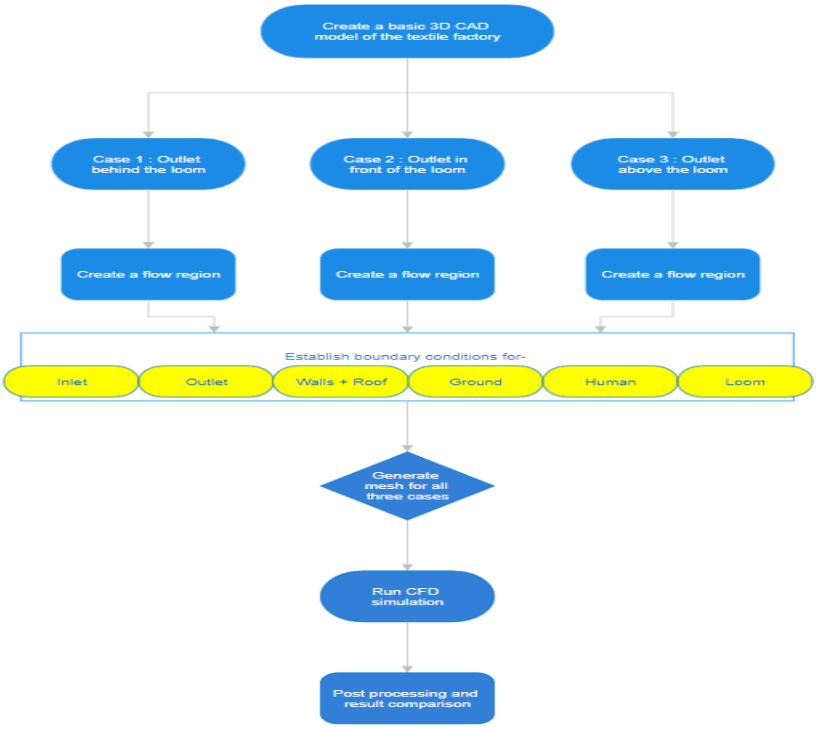

Fig. 1. Flowchart of the study

A. Geometry-

Basic floor plan of the model based on a small textile factory was created in Autodesk Revit 2024. A 6m x 6m x 6m room was first created. Later, a loom, and a human model were added into the project. Since the outside air flowing in the factory is uncontrolled as well as contain moisture and impurities, which are harmful for the textile production, the air needs to be conditioned. For this an inlet of dimension 16mm x 10mm was added into the system

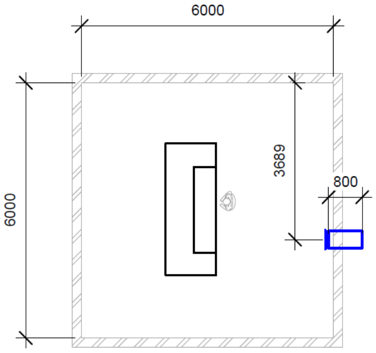

Fig.2. Basic floor plan with inlet position (in blue)

To optimise the ventilation system, the impact that the position of outlet has on the air distribution in the room is to be studied in this study. For this a ceiling mounted outlet of dimensions 24mm x 24mm face and 12mm x 12mm connection was added to the model. Three different cases were prepared for the study with three different positions of outlet. For case 1 the outlet is added at the right corner furthest away from the wall mounted inlet and behind the loom and human. For case 2 the outlet is placed at the right corner closest to the wall mounted inlet and in front of the loom and the human. For case 3 the outlet is placed at the centre of the room above the loom and the human. After placing the outlets at three different positions, three different models were prepared for further study. Fig. 3 show the three-3D models of the three cases which are used for CFD simulation.

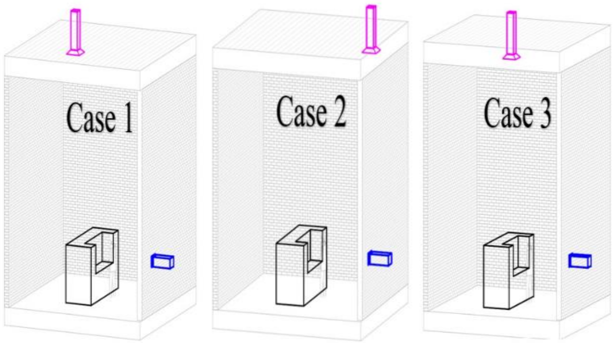

Fig. 3. 3-D model of three cases

B. Flow region–

Once the geometry is ready certain operations need to be performed over it to ensure that a continuous flow region is achieved within one volume. For this reason, using

SIMSCALE cloud CFD simulation internal flow region was created for the three cases as shown in figures 4. After the creation of flow region, the different parts of the geometry are deleted and only the flow region is taken forward for mesh creation.

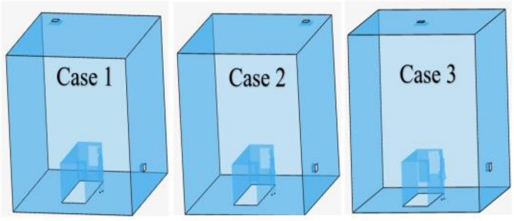

Fig. 4. Flow regions of three cases

C.**Mesh-**

Once the flow region is defined then in the next step a mesh needs to be prepared to establish the boundary conditions and the physics of the model. Mesh of the three models is created using the SIMSCALE cloud CFD simulation. A hex-dominant algorithm is adopted for mesh formation. To optimise and simplify the mesh generation- region refinement, inflate boundary layer and feature refinement measures are adopted. The goal is to ensure that the mesh quality is within the acceptable range of 0.035 to 1.0.

For case 1 mesh quality of 0.500595, case 2 mesh quality of 0.471934, and for case 3 mesh quality of 0.595831 are generated which is within the acceptable range. Fig. 5 gives the three meshes generated.

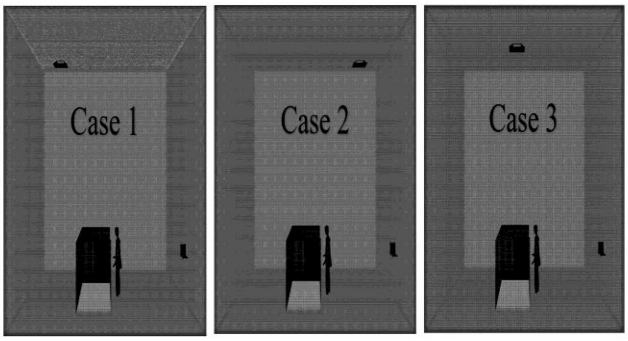

Fig. 5. Mesh generated for three cases

D.**Materials-**

Since the goal of the study is to ensure thermal comfort inside the room hence for the simulation only air is considered and all other impurities inside the air are neglected. The properties of the air are as shown in table 1. The air is made to flow inside the room from the inlet and later removed out via the outlet.

Table -1 Air properties

Viscosity model	**Newtonian**
Kinematic viscosity (ν)	**0.000015295 m2/s**
Density (ρ)	**1.196 kg/m3**
Thermal expansion coefficient	**0.00343 1/K**
Reference temperature (T_0)	**273.15 K**
Laminar Prandtl number (Pr_{lam})	**0.713**
Turb. Prandtl number (Pr_t)	**0.85**
Specific heat (c_p)	**1004 J/(kg-K)**

E. **Boundary conditions-**

Velocity Inlet- This condition is used to define the flow of air into the room. Air with fixed velocity of 3m/s (Ux = -3 m/s) and temperature of 298.15 K entered the system through the inlet as positioned in the model.

Pressure Outlet- This condition is used to define the flow of air out of the room. A pressure of 0 Pa mean value was defined at the outlet as positioned in the model. This creates a pressure gradient which is responsible for the suction of air out of the room.

Walls- This condition is used to define the radiation effect of the walls (including roof) of the room. Table 2 gives the properties defined under this condition.

Table-2 Wall boundary conditions

Velocity (U)	**No-slip**
Turbulence wall	**Wall function**
Temperature type	**External wall heat flux**
Heat flux	**Derived heat flux**
Heat transfer coefficient	**5 W/(K-m2)**
Ambient temperature	**303.15 K**
Wall thermal	**No wall thermal**
Initial boundary temperature	**303.15 K**
Radiative behaviour	**Opaque**
Emissivity	**0.9**

Ground- This condition is used to define the radiation effect of the floor of the room. Table 3 gives the properties defined under this condition.

Table-3 Ground boundary conditions

Velocity (U)	**No-slip**
Turbulence wall	**Wall function**
Temperature type	**External wall heat flux**
Heat flux	**Derived heat flux**
Heat transfer coefficient	**2 W/(K-m2)**
Ambient temperature	**298.15 K**
Wall thermal	**No wall thermal**
Initial boundary temperature	**298.15 K**
Radiative behaviour	**Opaque**

Human- This condition is used to define the metabolic heat produced by the humans. Assuming only one person to operate on the loom table 4 gives the properties defined under this condition.

Table-4 Human boundary conditions

Velocity (U)	**No-slip**
Turbulence wall	**Wall function**
Temperature type	**Turbulent heat flux**
Heat source	**Flux heat source**
Heat flux	**100 W/m2**
Initial boundary temperature	**307.15 K**
Radiative behaviour	**Opaque**
Emissivity	**0.9**

Loom- This condition is used to define the heat that the loom will produce either due to friction of threads or moving of loom parts. In addition, as the automated looms works on electricity, additional thermal effect must be considered. Table 5 gives the properties defined under this condition.

Table-5 Loom boundary conditions

Velocity (U)	**No-slip**
Turbulence wall	**Wall function**
Temperature type	**Turbulent heat flux**
Heat source	**Flux heat source**
Heat flux	**150 W/m2**
Initial boundary temperature	**313.15 K**
Radiative behaviour	**Opaque**
Emissivity	**0.9**

F. **Simulations-**

A SIMSCALE CFD simulation is run using a convective heat transfer analysis with k-omega SST as the turbulence model and a steady state time dependency. To incorporate the heat produced by the human and loom, radiation effect is also included in the model. No passive species are included in the model as all other impurities are neglected. The effect of gravity is incorporated in the model by giving a global gravity value of 9.81 m/s2 in the negative Y- direction. Simulation is run for a total of 2500 seconds with delta (iterations) value of 1 seconds. Every 500 seconds the results are saved and the Scotch decompose algorithm is adopted.

G**. Thermal comfort-**

To determine the thermal comfort level a total of nine points and six points are identified on the simplified loom model and the human model respectively. The figure 6 and 7 show these points, A-I on loom model and A-F on human model respectively. The temperature at these points is recorded for every 500 seconds of the simulation run and accordingly plotted on the chart to better analyse the result.

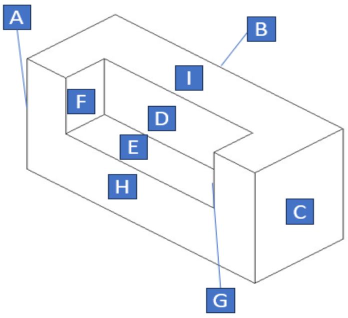

Fig.6. Simplified loom model's thermal measurement points

Once the temperatures of these points are recorded then to compare between the three cases the mean temperature of every point for different time interval is calculated and with the value of the mean temperature chart is prepared for easier comparison among the different cases. The case with the lowest temperature trend denotes the best thermal comfort case and that ventilation system will give the optimum result.

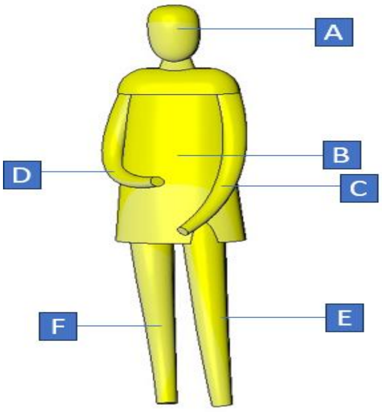

Fig.7.Human model's thermal measurement points

III. EXPERIMENT AND RESULT

A. **Case 1-**

From the experimentation it was seen that the conditioned air from the inlet is unable to lower the temperature of the loom and the human from the initial 313.15 K and 307.15 K respectively. Instead, the temperature rose to approximately 327 K initially at some regions and later stabilized around 324 K. Although the temperature of the loom and the human is managed but it is still higher compared to the desired temperature of 298.15 K. Figure 8 gives the temperature of the model after 2500 seconds of the simulation.

In addition, the temperatures at the identified points for the loom model as well as the human model are given in the table 6 and 7 respectively.

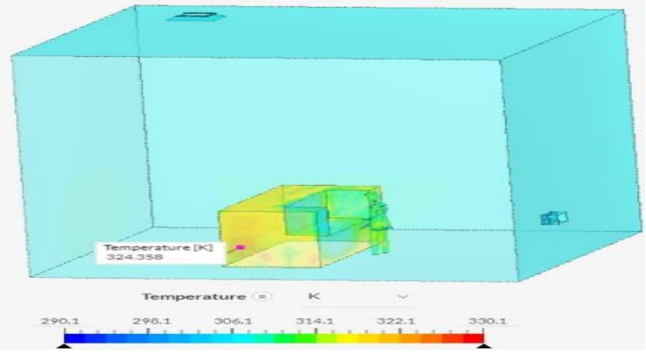

Fig.8. Temperature for case 1 after 2500 seconds

Table -6 Case 1 Loom model temperatures

Locations	Temp in K at 500 s	Temp in K at 1000 s	Temp in K at 1500 s	Temp in K at 2000 s	Temp in K at 2500 s
A	**320.752**	**315.22**	**316.862**	**317.567**	**316.828**
B	**317.393**	**316.421**	**317.143**	**317.066**	**316.631**
C	**311.266**	**314.951**	**313.234**	**317.159**	**315.689**
D	**303.146**	**301.65**	**300.955**	**301.19**	**301.378**
E	**308.833**	**302.267**	**303.247**	**302.834**	**303.675**
F	**301.012**	**302.358**	**302.267**	**303.808**	**304.248**
G	**316.141**	**304.849**	**305.2**	**304.968**	**305.256**
H	**309.99**	**302.703**	**305.706**	**304.676**	**307.547**
I	**316.993**	**317.541**	**317.541**	**315.708**	**315.649**
Avg	**311.725**	**308.662**	**309.128**	**309.442**	**309.656**

Table -7Case 1Human model temperatures

Locations	Temp in K at 500 s	Temp in K at 1000 s	Temp in K at 1500 s	Temp in K at 2000 s	Temp in K at 2500 s
A	**309.429**	**313.093**	**313.291**	**310.916**	**313.103**
B	**309.459**	**310.893**	**314.077**	**313.388**	**315.196**
C	**301.314**	**303.341**	**301.475**	**301.08**	**301.285**
D	**302.07**	**302.702**	**304.544**	**305.531**	**304.862**
E	**305.031**	**311.357**	**308.829**	**305.522**	**306.567**
F	**314.029**	**315.514**	**314.867**	**315.29**	**314.732**
Avg	**306.889**	**309.483**	**309.514**	**308.621**	**309.291**

Based on the obtained temperature values the corresponding charts are prepared

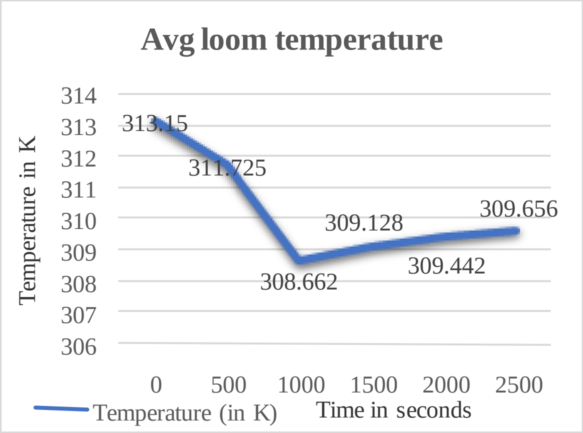

Fig. 2. Plot of avg loom temperature v/s time in case 1

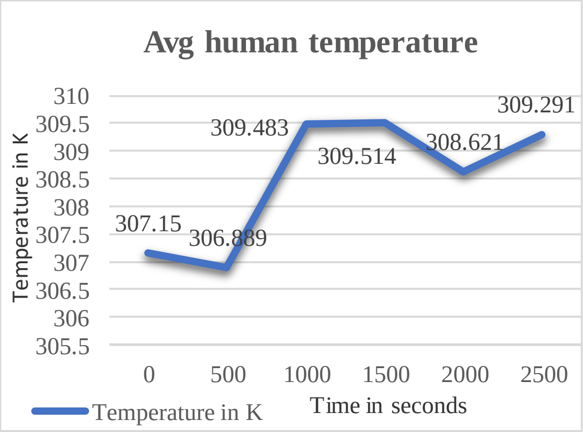

Fig.10. Plot of avg human temperature v/s time in case 1

B. **Case 2-**

The conditioned air from the inlet was unable to lower the temperature of the loom and the human from the initial 313.15 K and 307.15 K respectively. Instead, the temperature rose to approximately 328 K initially at some regions and kept on increasing to approximately 333 K at the end of the simulation run. Fig. 11 gives the temperature of the model after 2500 seconds of the simulation.

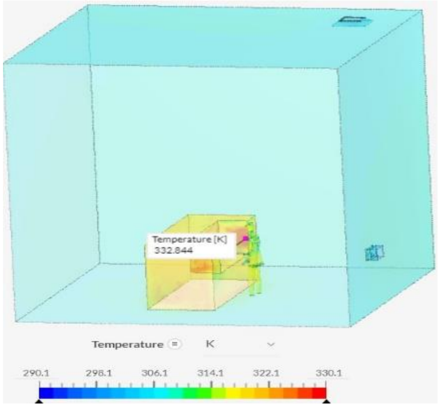

Fig.11. Temperature for case 2 after 2500 seconds

Table -8 Case 2 Loom model temperatures

Locations	Temp in K at 500 s	Temp in K at 1000 s	Temp in K at 1500 s	Temp in K at 2000 s	Temp in K at 2500 s
A	**316.727**	**321.717**	**323.429**	**321.617**	**321.737**
B	**315.484**	**318.898**	**316.74**	**318.464**	**318.944**
C	**313.135**	**315.165**	**315.525**	**316.485**	**313.813**
D	**316.368**	**317.943**	**318.726**	**319.419**	**319.898**
E	**320.438**	**320.7**	**321.078**	**321.475**	**321.303**
F	**325.556**	**321.371**	**329.815**	**329.116**	**328.842**
G	**327.608**	**329.761**	**329.271**	**331.808**	**329.226**
H	**316.92**	**319.155**	**318.639**	**318.157**	**317.281**
I	**314.965**	**315.965**	**317.766**	**316.635**	**317.196**
Avg	**318.578**	**320.075**	**321.221**	**321.464**	**320.916**

Table -9 Case 2 Human model temperatures

Locations	Temp in K at 500 s	Temp in K at 1000 s	Temp in K at 1500 s	Temp in K at 2000 s	Temp in K at 2500 s
A	**315.014**	**314.268**	**313.916**	**316.902**	**316.902**
B	**318.696**	**319.039**	**320.435**	**319.732**	**320.248**
C	**300.542**	**302.477**	**302.802**	**303.423**	**303.434**
D	**302.486**	**304.482**	**305.342**	**305.541**	**306.378**
E	**313.388**	**312.57**	**313.895**	**317.393**	**315.917**
F	**316.479**	**317.535**	**317.468**	**319.472**	**319.338**
Avg	**311.101**	**311.729**	**312.31**	**313.744**	**313.703**

Based on the obtained temperature values the corresponding charts are prepared.

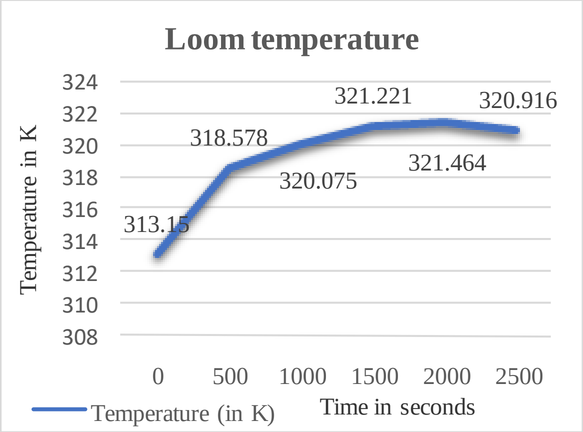

Fig. 12. Plot of Avg. loom temperature v/s time for case 2

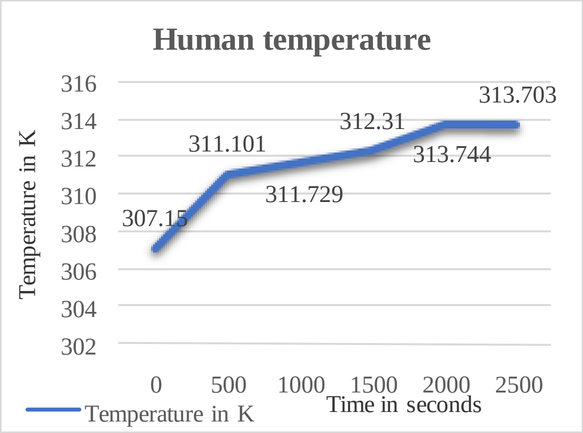

Fig. 13. Plot of Avg. human temperature v/s time for case 2

C. **Case 3-**

The conditioned air from the inlet was able to lower the temperature of the loom and the human from the initial 313.15 K and 307.15 K respectively to the desired 298.15 K at some regions. The air circulation around the loom and the human was able to maintain the desired temperature by the end of the simulation run. Fig. 14 temperature of the model after 2500 seconds.

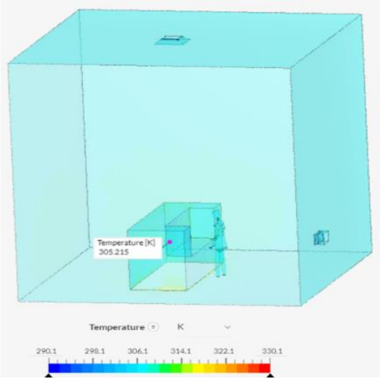

Fig. 14 Case 3 Temperature after 2500 seconds

In addition, the temperatures at the identified points for the loom model as well as the human model are given in the table 10 and 11 respectively.

Table -8 Case 3 Loom model temperatures

Locations	Temp in K at 500 s	Temp in K at 1000 s	Temp in K at 1500 s	Temp in K at 2000 s	Temp in K at 2500 s
A	**306.394**	**304.597**	**305.462**	**302.801**	**303.679**
B	**302.974**	**303.686**	**305.962**	**304.675**	**304.771**
C	**302.805**	**304.374**	**306.684**	**304.815**	**307.07**
D	**299.25**	**299.546**	**299.735**	**300.023**	**301.026**
E	**300.154**	**300.242**	**300.479**	**300.872**	**300.585**
F	**299.717**	**300.201**	**300.828**	**300.34**	**301.065**
G	**302.863**	**301.223**	**300.614**	**301.805**	**302.297**
H	**300.414**	**300.639**	**300.711**	**301.56**	**301.385**
I	**302.332**	**302.743**	**303.887**	**303.951**	**303.899**
Avg	**301.878**	**301.917**	**302.707**	**302.316**	**302.864**

Table- 11 Case 3 Human model temperatures

Locations	Temp in K at 500 s	Temp in K at 1000 s	Temp in K at 1500 s	Temp in K at 2000 s	Temp in K at 2500 s
A	300.925	301.263	301.229	302.613	303.036
B	301.022	302.602	302.587	303.147	303.768
C	300.263	300.749	301.251	301.857	302.003
D	301.294	302.675	304.571	304.221	304.954
E	299.754	300.827	301.261	301.365	302.177
F	300.666	302.322	302.389	303.104	303.161
Avg	300.654	301.74	302.215	302.718	303.183

Based on the obtained temperature values the corresponding charts are prepared.

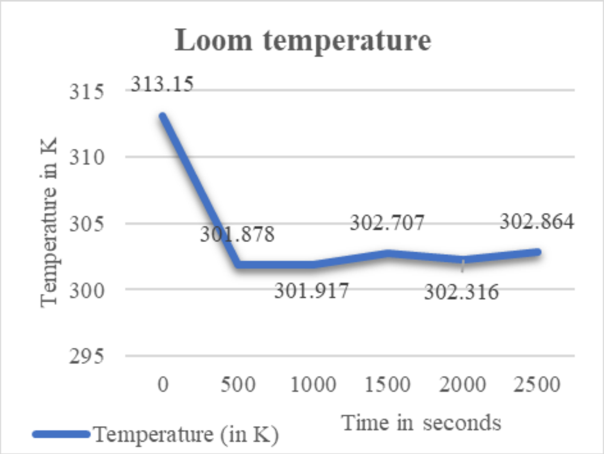

Fig.3. Plot of Avg. loom temperature v/s time for case 3

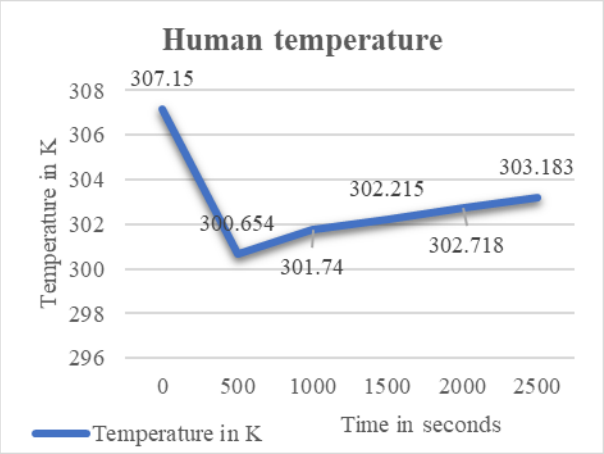

Fig.4.Plot of Avg. human temperature v/s time for case 3

IV.CONCLUSION

On comparing the air circulation within the domain, case 3 gives the best air circulation. This naturally will result in good indoor air quality. The fig. 17 and 18 give the loom and human temperature respectively for the three different cases. On comparing the temperature of loom and the human, case 3 again gives the best result as the temperature was able to reach and be maintained at the desired level. Finally, the comparison of loom temperature and human temperaturevariation with time has been performed. This will result in better thermal comfort for the workers as well as good operating temperature of the loom.

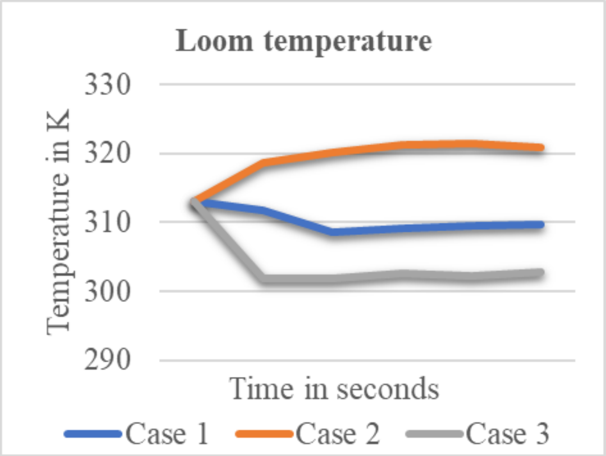

Fig.5. Loom temperature v/s time comparison for three cases.

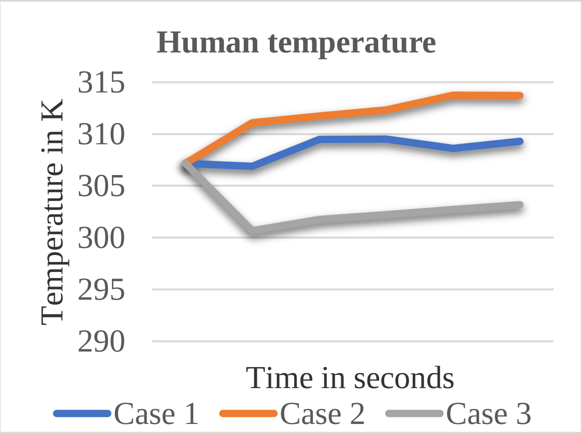

Fig. 6.Human temperature v/s time comparison for Three cases

On observing the variations in the results of the three different cases it can be concluded that the location of the outlet of the ventilation system in a room influences the overall efficiency of the ventilation system and it must be taken into consideration when designing a ventilation system. Thus, it can be concluded that the best ventilation system position out of the three cases is the Case 3 where the ceiling mounted outlet was placed at the centre of the room directly overhead the loom. This position of the ceiling mounted outlet can give better indoor air quality as well as better thermal comfort which will ultimately result in better quality of the fabrics as well as minimum breakdown of the loom due to heat i.e., lower maintenance time and cost.

V.FUTURE SCOPE

This study shows that the position of the air outlet as well as the location of the heat radiating machine and the working area has an impact on the air distribution in a small factory. Thus, more studies can be performed to establish a standard optimised manual for ventilation system installation in small industries. In addition, the impact of humidity on indoor air circulation was ignored in this study. This however cannot be possible in cases where humidity at certain level is need as in the cases of certain fabrics. In this study only single worker was introduced in the domain which may not be possible in real conditions and thus further studies may be required to understand the impact that multiple workers have on the ventilation system in a small industry. Anyhow the results of this study can be used as the basis for further simulations.

VI. REFERENCE

[1] Lakhal, A., et al. (2021). Influence of environmental factors on the working rhythm in a clothing industry. Industria Textila, 72(1), pp.108-113.

[2] Pollitt, J. (1950). The importance of humidity in textile manufacture. Journal of the Textile Institute Proceedings, 41(1), pp.1-6.

[3] Dixit, S. (2015). Enhancing the competitiveness of textile industry in an emerging economy: The role of MSME. SAMVAD, 9, pp. 26-39.

[4] Walker, A. (2010). Natural ventilation. Whole Building Design Guide: A Program of the National Institute of Building Sciences, pp.25-34.

[5] Elhadary, M. I., et al. (2021). A comparative study for forced ventilation systems in industrial buildings to improve the workers' thermal comfort. Sustainability, 13(18), pp.102-107.

[6] Meng, X., et al. (2020). Experimental study on the performance of hybrid buoyancy-driven natural ventilation with a mechanical exhaust system in an industrial building. Energy and Buildings, 208, pp.104-109.

[7] U.S. Department of Energy. (2020). Whole-house ventilation, retrieved from the article, https://www.energy.gov./energy.saver/weatherize report /ventilation/whole-house-ventilation)

[8] Miqdad, A., et al. (2017). Development of system to control air conditioner's airflow for spot cooling. 2017 International Conference on Engineering Technology and Technopreneurship (ICE2T). IEEE.

[9] Li, Y., & Nielsen, P. V. (2011). CFD and ventilation research. Indoor Air, 21(6), pp.442-453.

[10] Sodja, J. (2007). Turbulence models in CFD, University of Ljubljana., pp.1-18.

[11] Atkinson, J. (Ed.). (2009). Natural ventilation for infection control in health-care settings.

[12] Fanger, P. O. (1977). Discomfort due to air velocities in spaces. Proceedings of Meeting of Commission B1, B2, E1, of Inst. Refrig.3(4).

[13] Awbi, H. B. (1983). Domestic warm air distribution with low-grade heat systems. Proceedings of the International Seminar on Energy Saving in Buildings, pp. 628-635.

[14] Holmes, M. J., & Sachariewicz, E. (1973). The effect of ceiling beams and light fittings on ventilating jets. HVRA.2(4).

[15] Awbi, H. B. (1989). Application of computational fluid dynamics in room ventilation. Building and Environment, 24(1), pp. 73-84.

[16] Etheridge, D. W., & Sandberg, M. (1996). Building ventilation: Theory and measurement, John Wiley & Sons,2(50).

[17] Nielsen, P. V. (1980). The influence of ceiling-mounted obstacles on the air flow pattern in air-conditioned rooms at different heat loads. Building Services Engineering Research and Technology, 1(4), pp.199-203.

[18] Chen, Q. (1996). Prediction of room air motion by Reynolds-stress models. Building and Environment, 31(3), pp. 233-244.

IJEAST

INTERNATIONAL JOURNAL OF ENGINEERING APPLIED SCIENCE AND TECHNOLOGY

ABOUT IJEAST

International Journal of Engineering Applied Science and Technology (IJEAST) is a peer-reviewed, open access journal that publishes high-quality research papers in the field of Engineering, Applied Science and Technology.

IJEAST aims to provide a platform for researchers, academicians, and professionals to share their innovative ideas, research findings, and practical experiences with the global scientific community.

FOCUS AREAS

- Engineering
- Applied Science
- Technology
- Innovation & Development
- Interdisciplinary Studies

PEER REVIEWED

All submissions are rigorously peer reviewed to ensure quality.

OPEN ACCESS

Free and unrestricted access to research for all.

GLOBAL REACH

Connecting researchers and professionals worldwide.

TIMELY PUBLICATION

We ensure a swift and efficient publication process.

For more information, visit our website
www.ijeast.com

INTERNATIONAL JOURNAL
OF ENGINEERING APPLIED SCIENCE
AND TECHNOLOGY

editor@ijeast.com
www.ijeast.com
India

2455-2143