| | |
|---|---|
| Project: | **CFD simulation(s) for Test Facility 1** |
| Client: | **FA-Leaflet Production Ltd.** |
| Report No: | **1.0** |
| Version No: | **1.0** |
| From: | **FabricAir Engineering Department** |

# CFD Report: 3D Premium

FabricAir A/S Sandvadsvej 2, 4600 Køge, Denmark
+45 56 65 21 10
fabricair.com
Document ID 052019-1535615-1

## 1. INTRODUCTION AND DELIMITATIONS

This report concerns the indoor air quality in terms of air movement and temperature in FA-Leaflet Production Ltd.'s structure, CFD simulations have been made to test if the suggested air dispersion solution provides sufficient air movement with proper induction and no uncomfortable drafts in the occupied zone.

Figure 1 below shows the suggested ventilations ducts and flow model(s) as placed in the studied space. The report includes simplified geometry of the space, description of ventilation conditions, Computational Fluid Dynamics (CFD) simulation and review of the results.

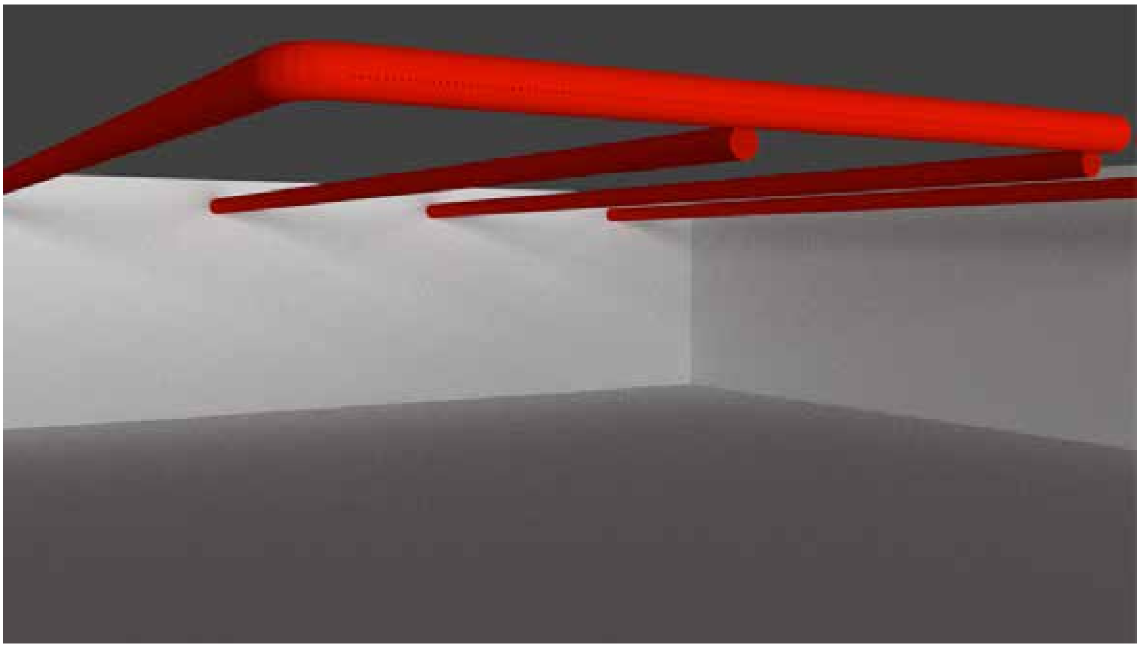

**Figure 1.** Visualization of room geometry and air duct placement (suspension elements are not included)

The purpose of this CFD report is to investigate the suggested air dispersion design in terms of:

1) Air movement

   a. Sufficient air movement and proper induction

   b. Less than 0.5 m/s air velocity in the occupied zone, with the majority of the air velocities below 0.25 m/s, creating a comfortable work zone.

   c. Airflow reaching the occupied zone in the area with the inclined ceiling.

2) Temperature

   a. Homogeneous temperature across the room with a gradient of max 1°C.

**Note:** The analysis provides a snapshot, taken in a split second in a "fluid" environment. The movement of the air is constantly stirring and a new snapshot in another split second will be different

## 1.1 ROOM GEOMETRY AND VENTILATION SYSTEM DESCRIPTION

The room is 54 m in length, 36 m in width and 9 m in height with an inclined ceiling on one end. For the purpose of running the CFD simulation(s), room geometry has been simplified to enable quicker simulations and still generate sufficient and valid results. Sophisticated forms are changed to simple geometrical shapes and the room is left completely empty. In doing so, we can increase the level of details concerning the flow models, hence achieve a more thorough result without delaying the process through unnecessary complexity in the calculations.

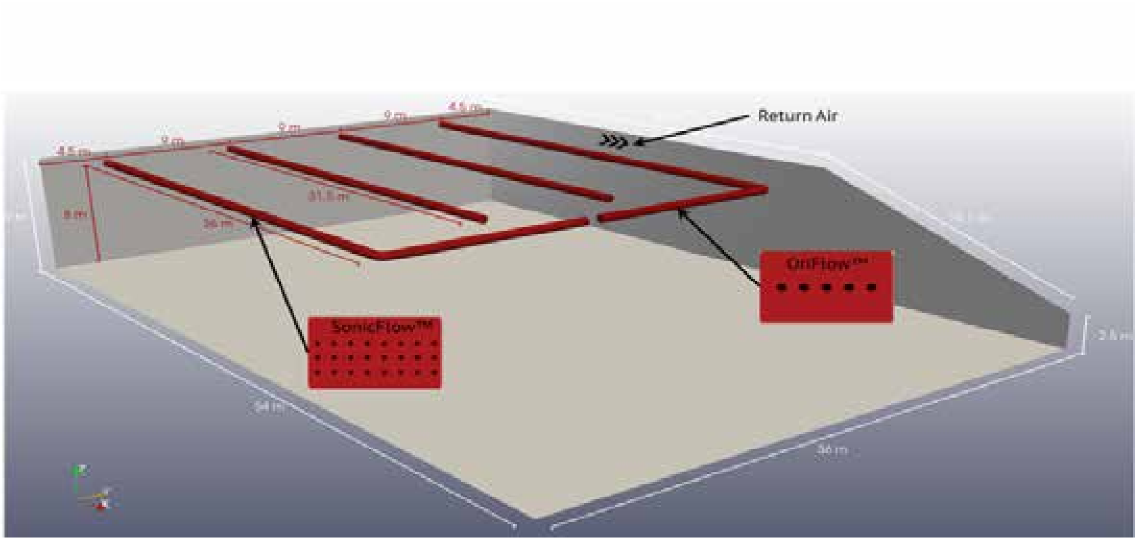


**Figure 2.** Simplifyed room geometry used for CFD simulations

The air dispersion system consists of four runs of Ø500 mm ducts in various lengths. The ducts are positioned in a symmetrical pattern; two of the ducts are longer and have a different flow model in the second half of the duct to supply air to the building area with the inclined roof. The distance between parallel ducts is 9 m. The ducts are suspended 8 m above the floor level (distance is measured from the floor to the middle of the duct).

Airflow is distributed through SonicFlow™ as calculated in project number 1535615 located at the 4 and 8 o'clock positions. The two longer ducts have OriFlow™ on the second half of the duct, after the 90° bends. Return air is located at the side walls and is not defined by FabricAir.

The air dispersion solution is set to a static 120 Pa pressure, with a total airflow of 20.000 $m^3/h$. The temperature of the supply air is 20° C which is 4° C lower than the room temperature of 24° C.

Boundary conditions are not included, neither are heat sources, such as heat radiation from walls, floor and ceiling.

**Disclaimer**

This report answers the questions as put forth in section 1. The CFD simulation is run according to the conditions described in the section 1.1 “Room geometry and Ventilation System Description” and section 1.2 “Description of CFD Simulation”.

Results are condition sensitive. This report is only valid for the specific FabricAir dispersion solution, based on the specific fabric type, weight of fabric, flow model(s), etc. Any change to these conditions could have significant influence on the results.

## 1.2 Description of CFD Simulation

CFD analysis is the most advanced method of predicting and evaluating an air dispersion system’s performance theoretically.

The main goal of this CFD simulation is to ensure the selected flow model(s) applicability to the project. Air should be dispersed evenly across the room with velocities in the occupied zone below 0.5 m/s, ideally around 0.25 m/s to prevent uncomfortable drafts in the work areas.

The first step in CFD modeling is converting 3D geometry into 3D mesh. The air medium is divided into finite number of volumes that are used by the CFD software to solve all parameters critical to the airflow. The size of the mesh volumes is refined according to the flow pattern. The cells in a path of flow are refined to be finer than the others to increase the accuracy of the simulation. The CFD program calculates all relevant flow parameters, such as air velocity, pressure, temperature, turbulence and so forth in each mesh cell.

The analysis is made after the room is fully balanced and the case is set up as an incompressible case. Boussinesq approximation was used for the thermal buoyancy, which means that insignificant variations in temperature, and thus in density, are ignored except in the buoyancy term to create a simpler, yet valid analysis.

## 2. CFD Simulation Results and Review

In the following section calculated air velocities are visualized using multiple slices located in the simulated volume. The slices’ location can be seen in the figure below: horizontal section view (HV), transverse section view (TV), longitudinal section view (LV), inclined section view (IV), air velocity diagram (D1) and probe(s) (P). The occupied zone, from floor to 1.8 m above floor level, is indicated using a red line in images where this is relevant.

The airflow velocities are visualized using a rainbow color scheme on a scale ranging from 0 to 0.5 m/s, where red represents air velocities at 0,5 m/s or higher and blue represents the lowest velocity at 0.05 m/s. Temperature is not visualized in the section views below.

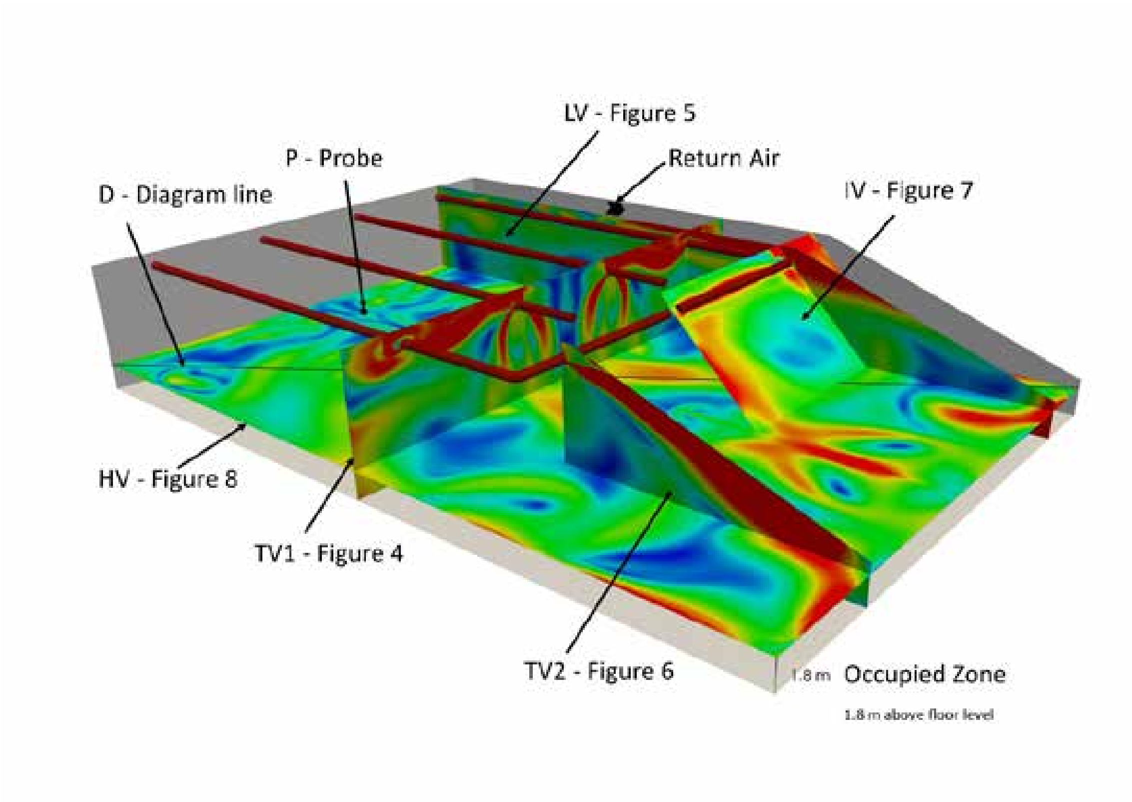


**Figure 3.** Location of the slices for analysis of air velocities

Figures 4 and 5 below show how air is discharged from the ducts via the SonicFlow™ flow model at the 4:00 and 8:00 o'clock positions on the duct.

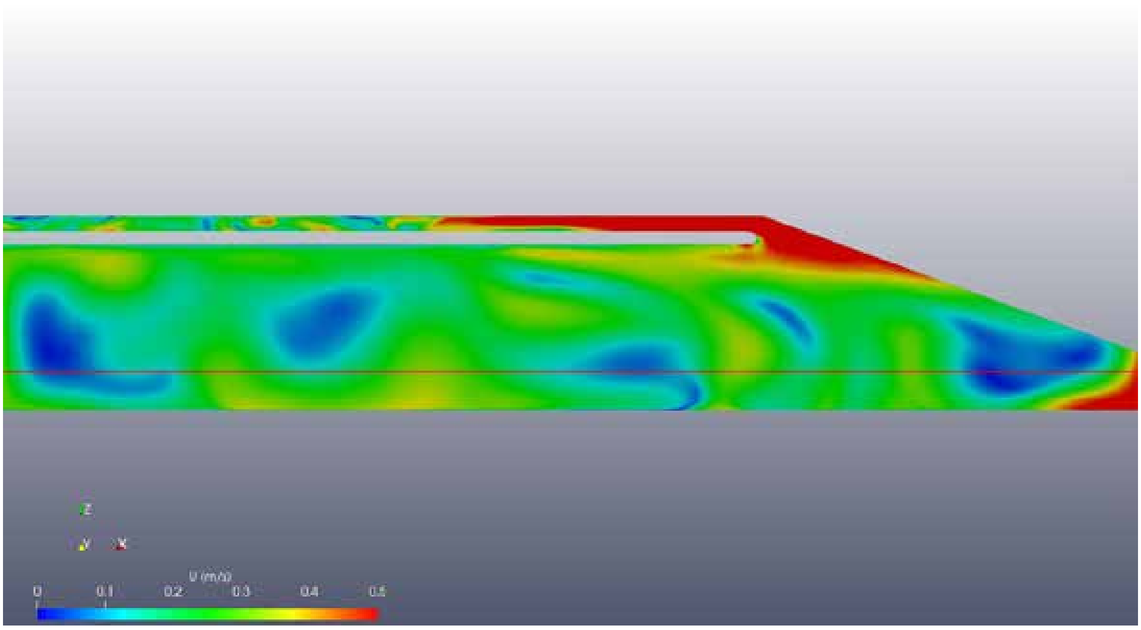


**Figure 4.** Transverse section view. Air velocities (scale 0-0.5 m/s).

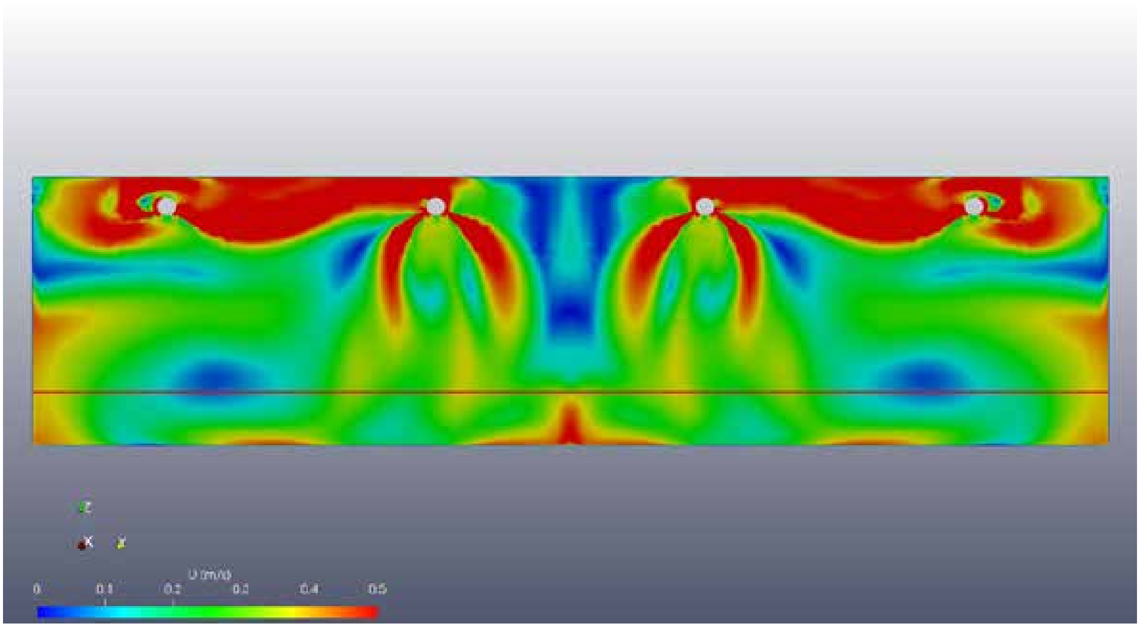

**Figure 5.** Longitudinal section view. Air velocities (scale 0-0.5 m/s).

In Figure 6 below, the part of the building with the inclined ceiling is shown. In this part of the building, the OriFlow™ flow model is used to supply air to the very end of the space, ensuring against stale air zones.

At this location air stream attaches to the ceiling forming stronger air stream. This effect helps to throw the air further without having to hang ducts in the area with the inclined roof.

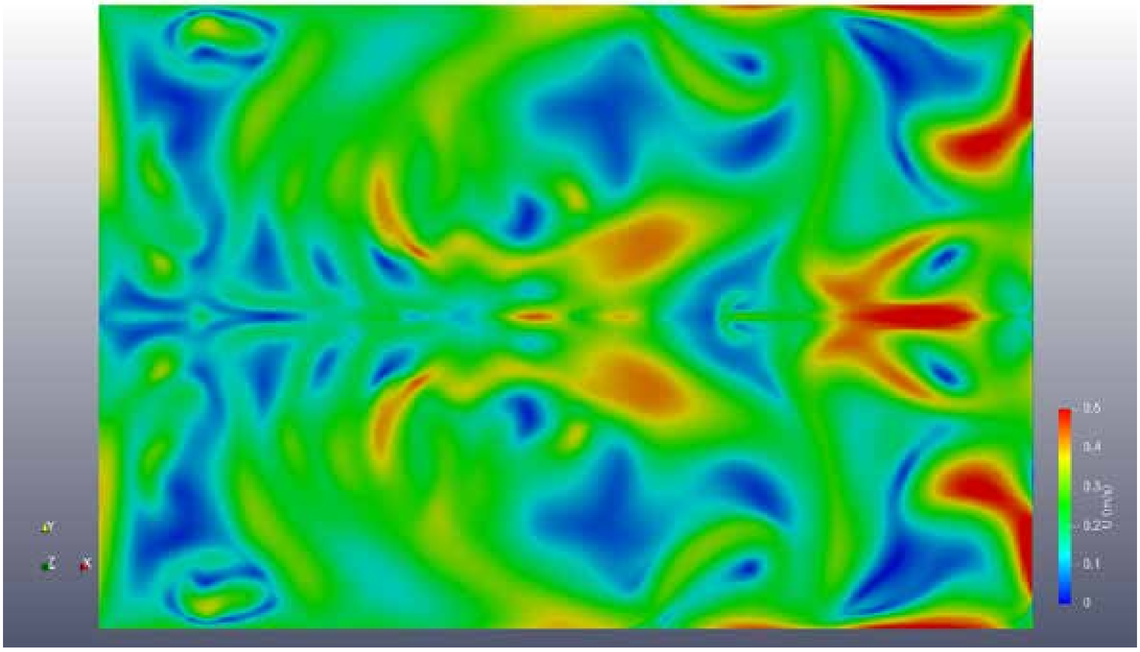

**Figure 6.** Transverse section view. Air velocities (scale 0-0.5 m/s).

Figure 7 below shows how airflows, discharged from the ducts via the SonicFlow™ and OriFlow™ flow models, affects the air movement in the occupied zone in the area of the building where the inclined ceiling begins.

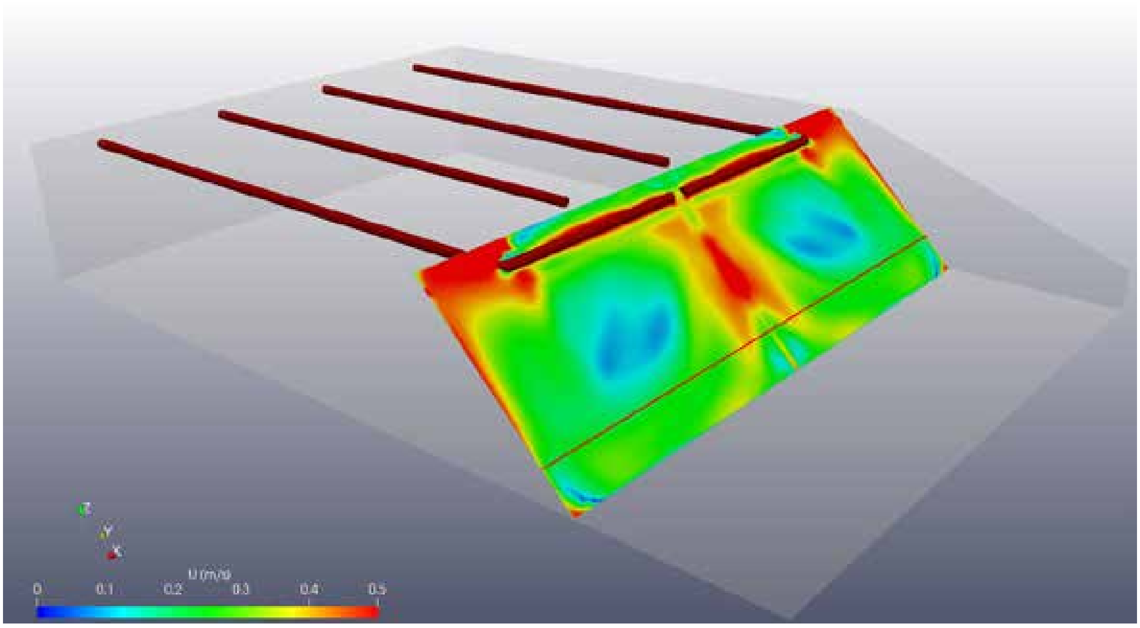


**Figure 7.** Inclined section view. Air velocities (scale 0-0.5 m/s).

Figure 8 shows a horizontal sectional cut, made at the height of the beginning of the occupied zone (1.8 m above the ground).

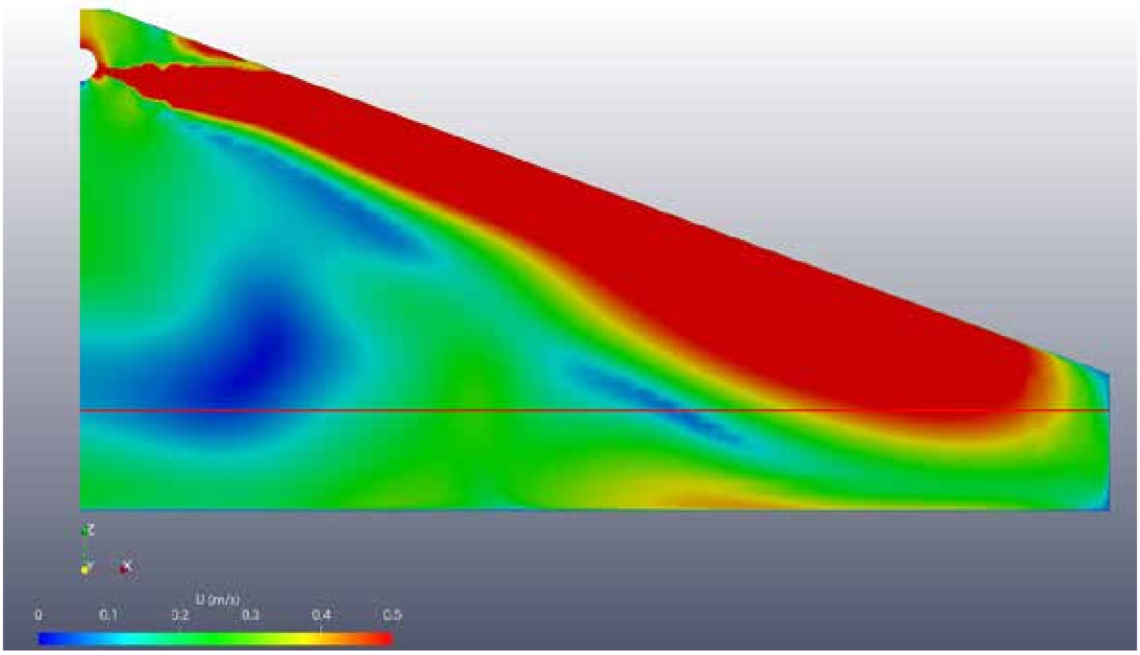


**Figure 8.** Horizontal section view. Air velocities (scale 0-0.5 m/s)

The majority of air velocity is below 0.5 m/s and mostly below 0.25 m/s. This is expected and within the question raised.

It will never be possible to avoid “red” zone 100%.

The diagram in figure 9 shows the air velocities’ distribution at the height of 1.8 m (the beginning of the occupied zone) on a diagonal from one corner of the room to the opposite corner (please see figure 3 for a visual explanation of the placement of this diagram line).

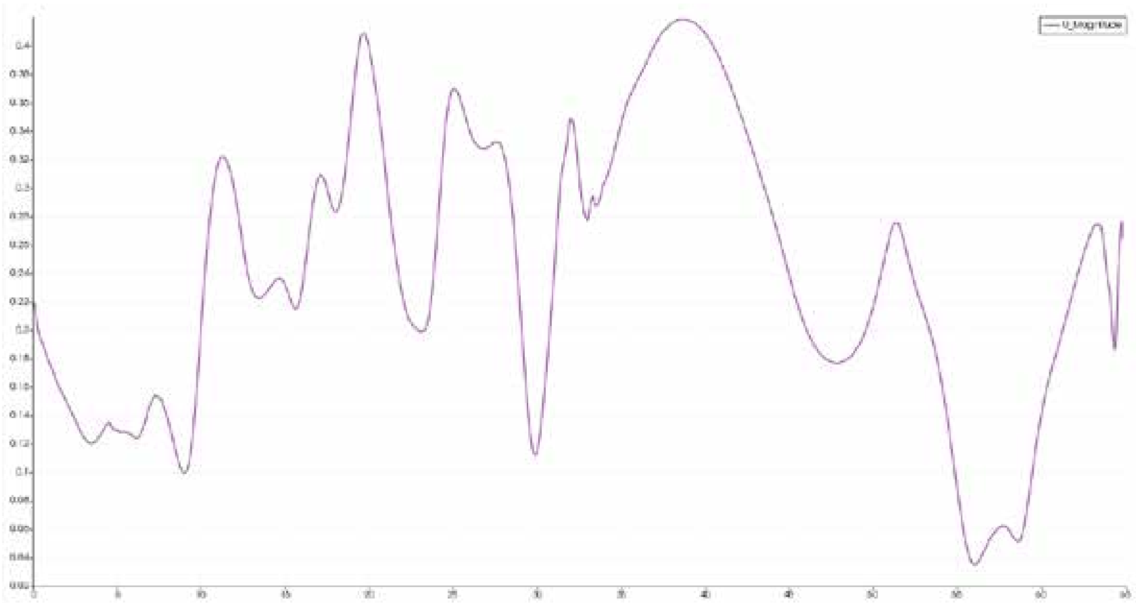

**Figure 9.** Air velocity diagram in 55 m distance along the diagonal of the room

At point 0 m (close to the wall) the air movement velocities are lowest due to effect of the wall. The walls’ influence can be seen on the velocities in the figure above, showing the 0-65 m range on the diagonal of the room. The average velocity in the occupied zone is 0.24 m/s.

Table 1 shows the results from the probe.

| | **Location** | | | U | T |
|---|---|---|---|---|---|
| | X | Y | Z | (m/s) | (K) |
| Probe 1 | 5 | 17 | 1.8 | 0.19 | 294 |

**Table 1.** Results from the probe

## 2.1 Overview in Vectors

In Figure 10 below, the airflows are shown as vectors. The vectors show how the air moves and the velocity is indicated by a rainbow color scheme. The scale ranges from 0 to 0.5 m/s, where red represents the highest air velocities at 0,5 m/s or higher and blue represents the lowest velocity at 0.05 m/s. The arrow tip is the direction of the air.

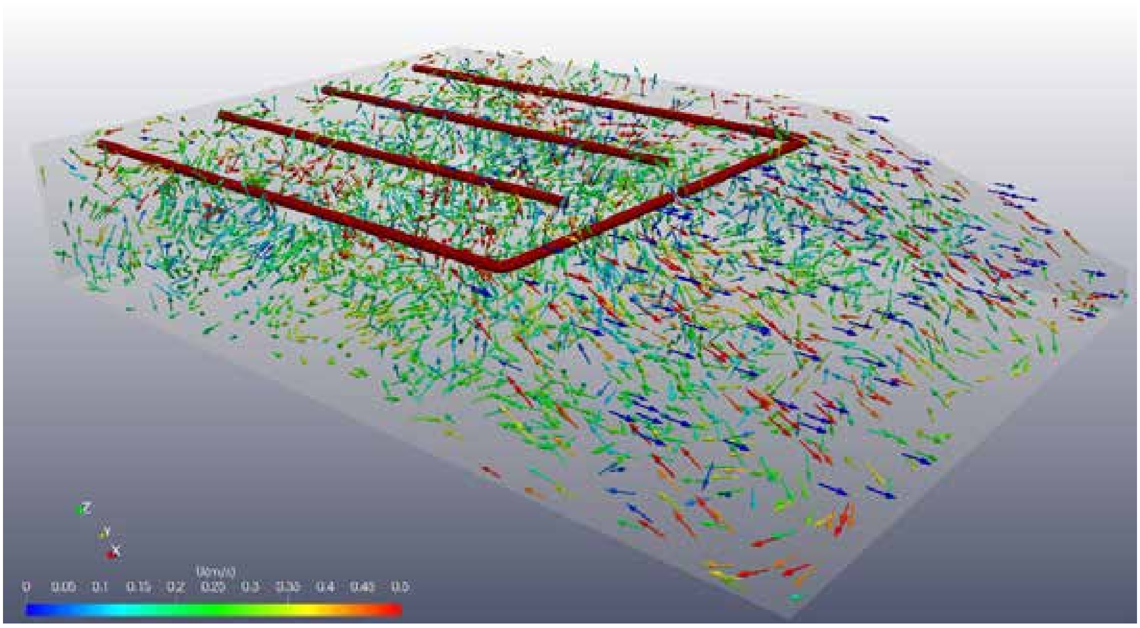


**Figure 10.** Circulation of air in the building

This image provides an overview of whole solution and shows proper mixing.

The CFD confirms that there is sufficient air movement. There is mixing of air in the entire room. Even though, the thermal convection lifts airflow up, the momentum that appears from 120 Pa static pressure pushes some of the air to the floor, ensuring a high indoor air quality in the occupied zone without drafts.

That the majority of the vectors are green, indicates good air movement and mixing without drafts. The figure shows an even air dispersion, where the airflow reaches all areas of the room.

## 2.2 Temperature Distribution

In the transverse section view in Figure 11 below, the temperature distribution (in Kelvin) is visualized using a traditional color scheme. The blue color range indicates supply air temperature and the red range indicating the original room air temperature in order to test whether the air mixing has resulted in a homogeneous temperature distribution. The white line indicates the occupied zone (1.8 m above the ground). The temperatures are shown after room is fully balanced to evaluate the full effect of the air dispersion solution.

The test confirms a homogeneous temperature across the space when the room is fully balanced. The original room temperate was 24° C. The temperature of the supply air is 20° C, which is 4° C lower than the room temperature. The transverse section view indicates an even temperature distribution across the space, with no areas that are particularly hot or cold. We know from the airflow analysis that there are no stale air zones either. The temperature gradient is below 1°C in the occupied zone. In this way no additional thermal discomfort would be felt.

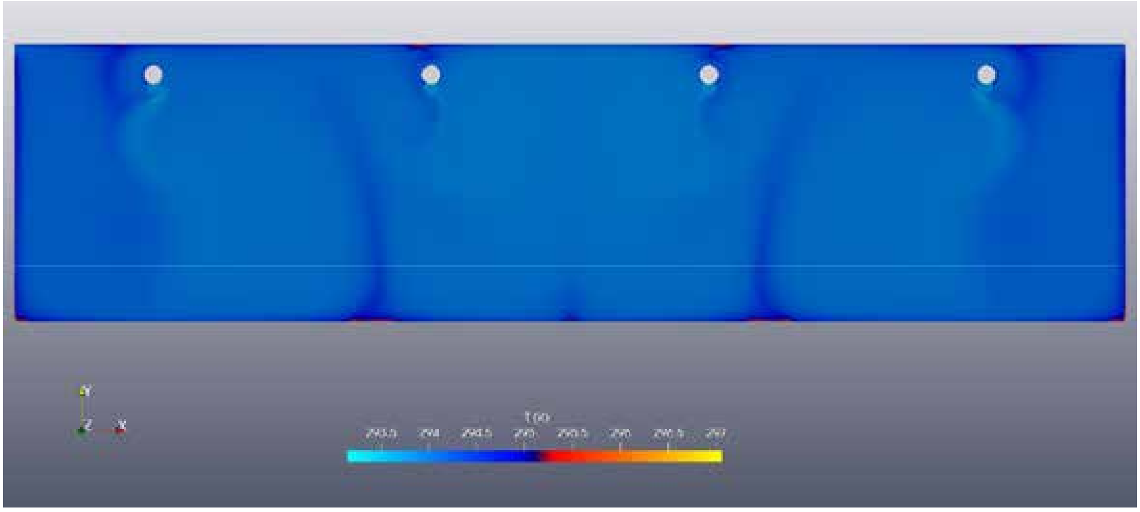

**Figure 11.** Transverse section view (TV1). Temperature distribution (scale 294-297K).

## Summary and Recommendations

A CFD simulation for FA-Leaflet Production Ltd.'s structure was performed. The focus of the CFD simulation was an evaluation of the parameter of a comfortable, draft-free work area, proper mixing and a homogeneous temperature (max 1*C). It should not be considered as complete study of the ventilation conditions of the building. Such a study would require a more detailed modeling with full geometry of the building, and boundary conditions.

The CFD simulations in this report are valid for the suggested FabricAir dispersion solution only and cannot be used as proof of similar results with comparable technologies.

The suggested air dispersion solution consisting of four Ø500 fabric ducts with a combination of the flow models SonicFlow™ and OriFlow™ was tested and the results of the CFD simulations confirm that the engineered solution in its current configuration is a good fit to the application under the given premises.

The solution ensures an even air dispersion across the room, and that the airflow reaches all areas of the room. The air velocities create sufficient air mixing in the occupied zone without generally getting higher than the recommended 0,5 m/s in the occupied one. In most of the occupied zone the air velocities stay below 0.25 m/s. There is a homogeneous temperature across the conditioned space.

In the area of the room with the inclined ceiling, the longer throw from OriFlow™ is suggested to ensure sufficient air movement to the very end of the space.

With an average air velocity in the occupied zone at 0.24 m/s, the air dispersion design ensures sufficient air movement and a pleasant draft-free environment without stale air zones. Even in the area with the inclined ceiling, the analysis confirms that the air movement is sufficient without creating uncomfortable drafts.

As less than 3% the total area has indications of a slight draft, the CFD confirms that the suggested design for the air dispersion solution is the right fit for the given space.

The temperature analysis confirms a homogeneous temperature across the space, with a temperature gradient of less than 1°C in the occupied zone. The suggested air dispersion design thus ensures that thermal comfort is achieved without drafts, resulting in a good indoor air quality.