Innovating the HVAC industry since 1973

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 Essential

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 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.

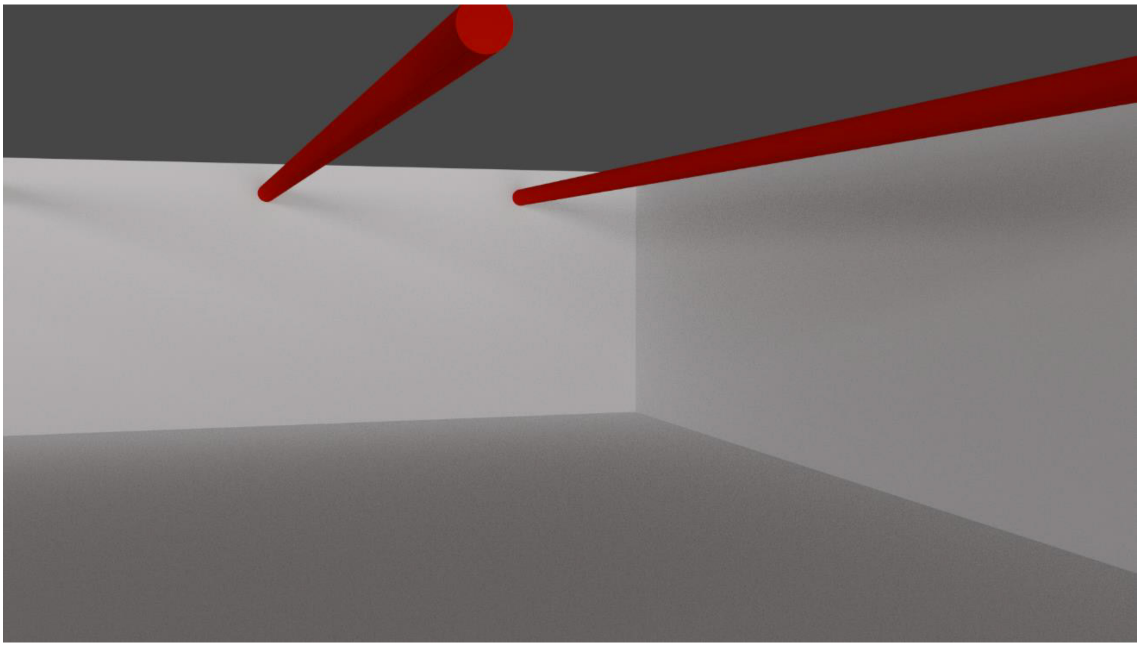


Figure 1. Visualization of room geometry and air duct placement (suspension elements are not included)

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.

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 36 m in length, 18 m in width and 9 m in height. 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. The CFD simulation is made only in the 1-meter section close to the back wall, as can been seen in Figure 2 below.

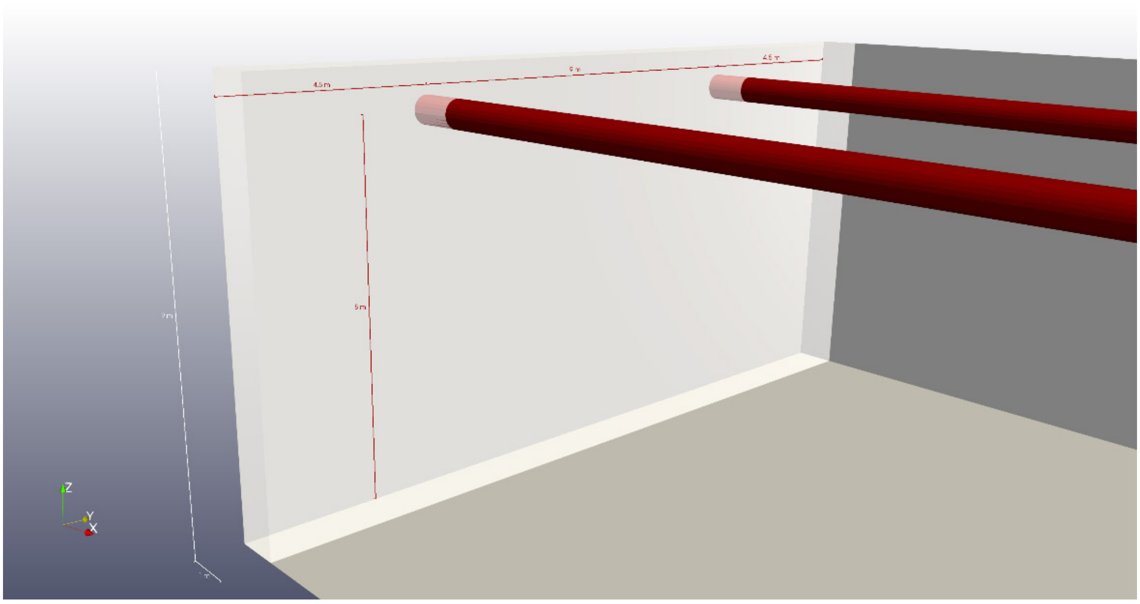


Figure 2. Simplified room geometry used for CFD simulations

The air dispersion system consists of two runs of Ø500 mm ducts 31,5 m in length. The ducts are positioned in a symmetrical pattern. The distance between parallel ducts is 9 m and the distance from the wall to the ducts is 4,5 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. 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 10.000 m^3/h (total airflow of the room). The temperature of the supply air is 20° C which is 4° C lower than the room temperature of 24° C.

External elements, such as 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 “Introduction and Delimitations”. 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 a transverse section view located in the simulated volume, which represents a specific slice of the full space. The slice’s location can be seen in the figure below. 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.

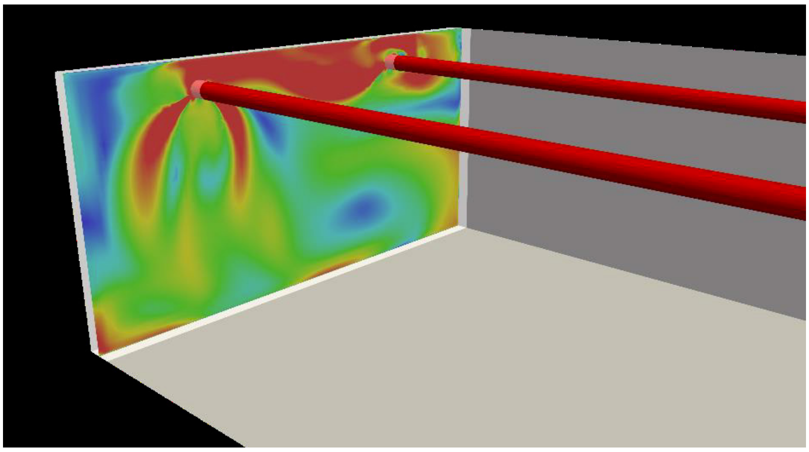
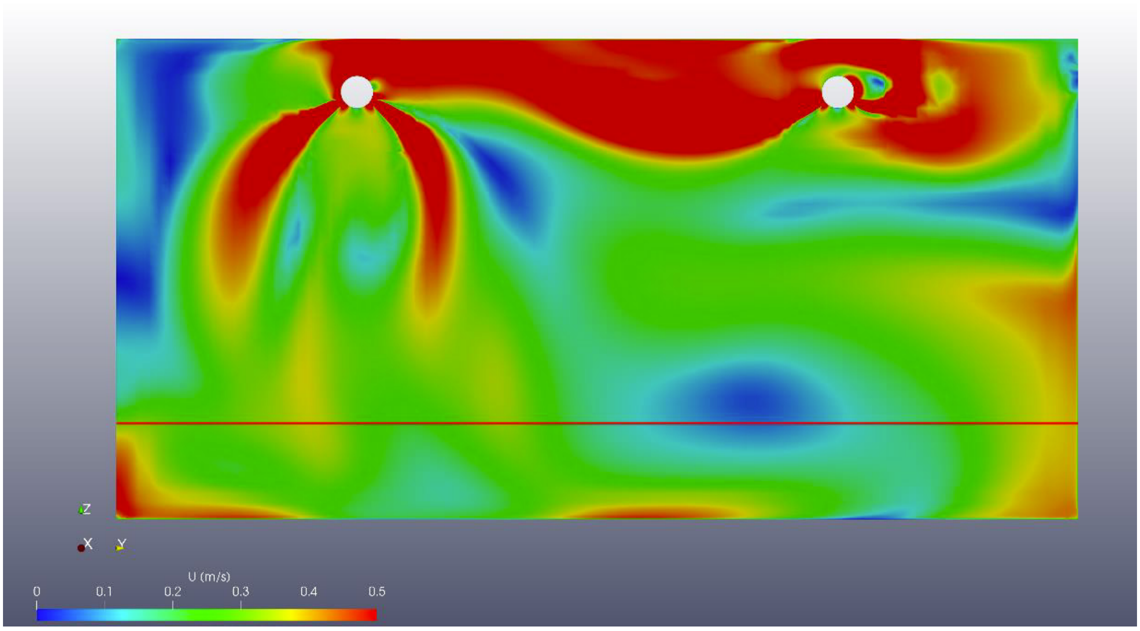


Figure 3. Location of the slice for analysis of air velocities



Figures 4 below shows 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.

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%.

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 and proper mixing. 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 external elements, such as 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 two Ø500 fabric ducts with the SonicFlow™ flow model, 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.

As indicated by the analysis of the room slice, 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. 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.