Mechartés
SIMULATION EXPERTS

CFD Modelling in Generator Room / DRUPS Room / Transformer Room

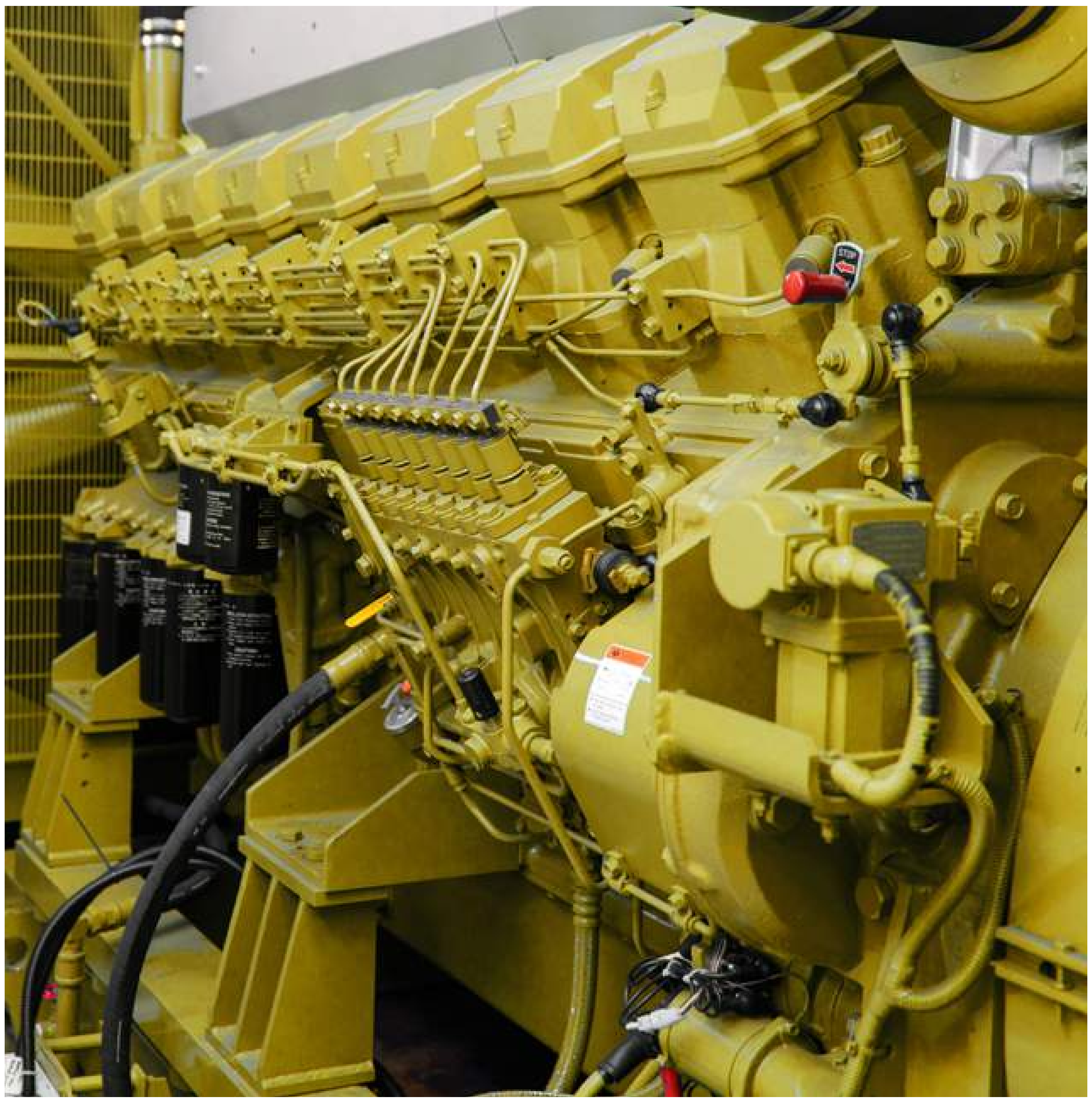
Table of Contents

Introduction

Uninterrupted electrical power supply is essential to the continuity of almost every business. Even a brief disruption in the power supply can impact normal operations and incur huge financial or even life losses. For instance, it is extremely critical in facilities like hospitals, data centers, etc.

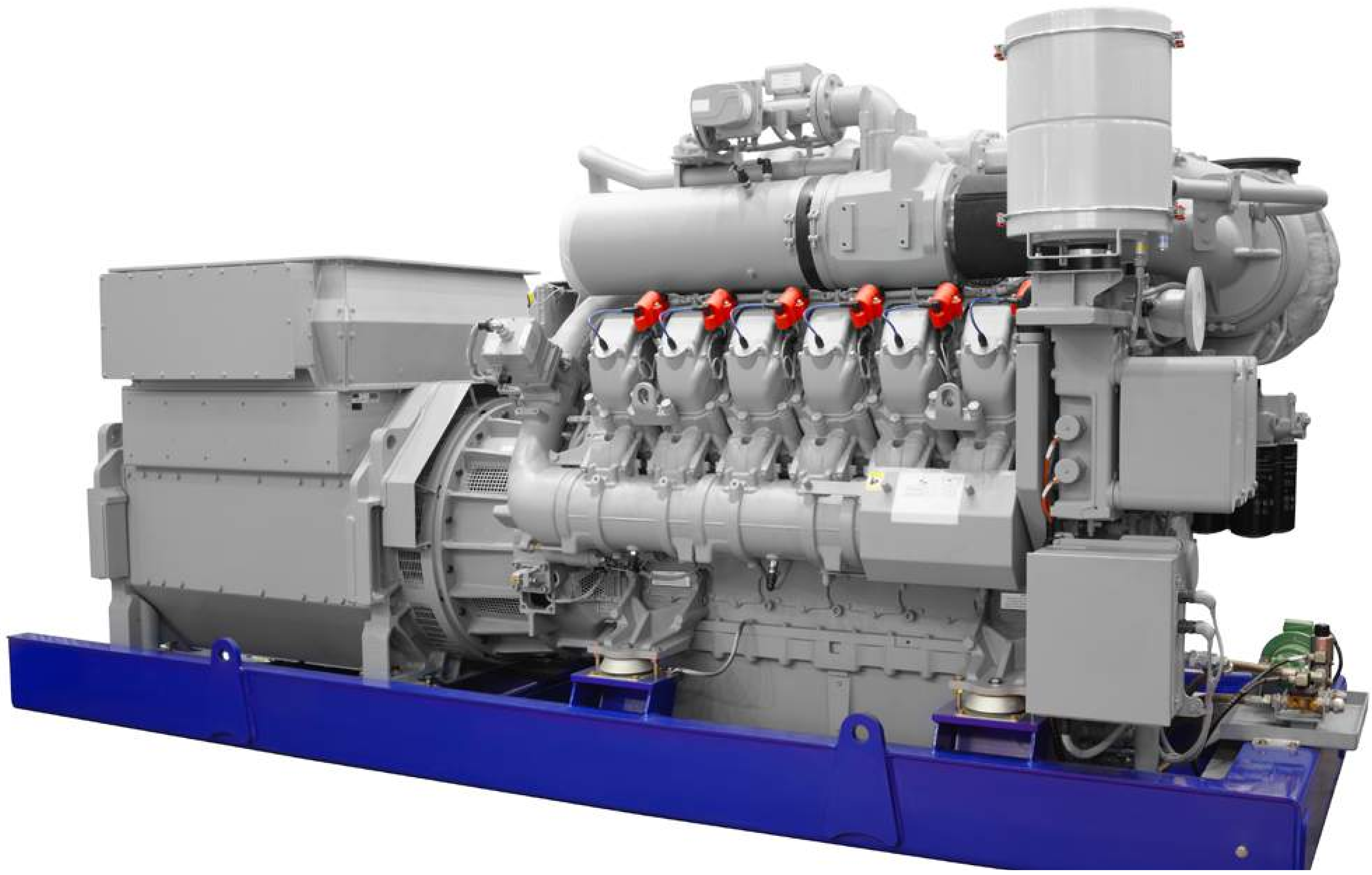
A backup Diesel Generator set (DG set) or Dynamic Rotary Uninterruptible Power Supply (DRUPS) are important alternatives for business owners which offers the ability to start and assume electrical load in a few seconds, providing power when the utility supply has failed. Therefore, many building owners and decision-makers install Diesel Generator sets (DG sets) as a substitute to supply their facilities during emergencies.

Backup DG sets are available in a wide range of capacities, ranging from kilowatts (KW) to megawatts (MW). Depending on the type of building requirement and space constraint, they can be installed within a building (at the rooftop or basement) or outside the building within customized DG enclosures.



Introduction

Diesel Rotary Uninterruptible Power Supply Devices (DRUPS) are slightly different than DG rooms. They combine the functionality of a battery-powered or flywheel-powered UPS and a diesel generator. When the main electricity supply fails, stored energy in the flywheel is released to drive the electrical generator, which continues to supply power without interruption. At the same time (or with some delay of a few seconds to prevent the diesel engine from starting at every incident), the diesel engine takes over from the flywheel to drive the electrical generator to make the electricity required.



The electro-magnetic flywheel can continue to support the diesel generator in order to keep a stable output frequency.

How Do You Ventilate a Generator Room (Fresh Air/Exhaust Air)?

How Do You Ventilate a Generator Room (Fresh Air/Exhaust Air)?

A well-designed generator room will ensure that:

- DG Sets / UPS / transformer sets are accessible.
- Manufacturer- and code-required clearances are maintained (fire protection regulations, etc.)
- Major components can be removed and replaced.
- Clean and relatively cool air can circulate around the generator set.
- Ventilation airflow (room inlet airflow) is adequate to reject the heat produced during operation and support the engine combustion process.
- Recirculation and bypass airflow is minimized, noise and vibration within and outside the building comply with code requirements, and ancillary components external to the generator set operate reliably.

Having a wide experience in simulation consultancy, Mechartes is the expert choice when it comes to Thermal flow and FEA-related simulation services.

How Do You Ventilate a Generator Room (Fresh Air/Exhaust Air)?

Ventilation of the genset room has two main purposes.

I. To avoid air recirculation to ensure that the performance of machines doesn't get affected, and

II. To provide an environment for the maintenance/ operation personnel so they can carry out their routine job comfortably.

In the genset room, right after the start of the machine, air circulation begins due to the radiator fan. Fresh air enters from the vent located behind the alternator, which passes over the engine and the alternator and cools down the engine body to a certain degree, and the heated air is discharged into the atmosphere through the hot air outlet located in front of the radiator.

For efficient ventilation, the air inlet/outlet opening should be of suitable dimensions. Louvers should be fitted to the windows to protect the air outlets. The louver fins should have openings of sufficient dimensions to make sure that air circulation is not being blocked. Otherwise, the occurring backpressure might cause the genset to overheat. Information about air inlet/outlet opening sizes and louver details should be obtained from a knowledgeable consultant as well as from the manufacturer.

The exhaust system (silencer and pipes) is installed to reduce the noise from the engine and to direct the toxic exhaust gases to appropriate areas. Inhalation of exhaust gases is a possible death hazard and should always be kept in mind while dealing with the exhaust system.

How Do You Ventilate a Generator Room (Fresh Air/Exhaust Air)?

The exhaust system should consist of a flexible compensator, silencer, and pipes that absorb vibration and expansion. Exhaust pipe elbows and fittings should be designed to accommodate expansion due to temperature.

The inlet and outlet air of the engine room should not be placed on the same wall to avoid short-circuiting the airflow and affecting the heat dissipation effect. However, if there is any difficulty, the air outlet should be on the upper side of the wall and the air inlet should be on the lower side.

In the cold area, attention should be paid to the influence of the air inlet and the air outlet on the temperature of the machine room, so as to avoid the low temperature in the machine room affecting the starting of the unit.

Challenges in Achieving Proper Ventilation Airflow and the Role of CFD/FEA Simulations

Challenges in Achieving Proper Ventilation Airflow and the Role of CFD/FEA Simulations

Key points in DG Rooms/DRUPS Room/Transformer room design which shouldn't be ignored and are equally critical for safe operation of the equipment and employees who need safe access to these enclosed spaces are:

- **Temperature:** Regulating the temperatures of DG rooms is important since engines and other electronic equipment can get quite hot while in use. A proper ventilation design with adequate source of air inlet and outlet will help keep the designated room at a safe temperature and prevent the equipment from overheating.
- **Airflow ventilation/Recirculation:** To have good air movement through vents, the use of fans/exhaust vents or air curtains is essential. It not only controls temperature but also ensures a steady flow of fresh air. This is important not just for effective working of the engines but also for any human presence in the room, which might be required for routine operation check-up or maintenance activity
- **Air Cleanliness:** A good, designed ventilation validated by CFD helps to remove harmful fumes and gases from these enclosed spaces. Engine exhaust and other harmful gases can be dangerous for the supporting staff if not checked during the design stage.
- **Noise:** Noise due to vibrations of these equipment or noise due to engine, alternator, engine exhaust or structural noise due can create unfavourable conditions for the occupants of the building and as per local building codes. With the help of simulations and as per

applicable codes, source of the noise can be identified and sound level (dB) can be checked by using effective sound barriers, insulators, isolation mounts, etc.

It is crucial to solve the problem of ventilation, smoke exhaustion, flue gas recirculation RH, noise reduction, and vibration reduction of the DG units, transformers, DRUPS placed in a room and the effect of ambient conditions on it. It should be analyzed and validated well in the design stage and as per applicable standards to avoid any future breakdown.

A CFD model is built upon fundamental physical equations of fluid flow and energy transfer. The technique is capable of providing time-dependent and as well as steady-state solutions to the coupled differential equations that govern fluid flows. Its key benefits are an ability to represent the effects of very complex geometries coupled with a means to solve complex flow problems based on more fundamental modeling of the physics involved.

Challenges in Achieving Proper Ventilation Airflow and the Role of CFD/FEA Simulations

CFD can help us to analyze the vital parameters in different operating conditions:

- No. of generator's unit operating at 100% load.
- Effect of temperature and airflow due to air intake louvers and exhaust air louvers (in open or close mode).
- Spacing required within the DG, transformers, DRUPS, or other related equipment for proper functioning and maintenance-related activity to be carried out without any hindrance.

Before implementing any design changes, analyze existing conditions within the facility with the help of specialized simulation experts like **Mechartes**.

DRUPS Room CFD Case Study

Objective:

The objective of this project was to examine the designed ventilation system effectiveness inside the "DRUPS room located on the ground floor of a Data Center at Abu Dhabi, UAE" using steady-state CFD analysis for the two scenarios. These two scenarios are:

- **Scenario-1 (4 units at 2000kw** in an independent mode): All the DRUPS units are working, and all the motorized dampers are open.
- **Scenario-2 (2 units at 2000kw** in an independent mode): Two DRUPS units are working and motorized dampers for the working unit will remain open. All other dampers will remain closed.

The specific aim of the CFD analysis is to check the following parameters for the designed ventilation system and given heat load conditions from DRUPS units:

- Whether the proper air circulation is maintained or not.
- Whether the average temperature of less than 55^0C is maintained or not in the DRUPS room.
- Whether the temperature at the alternator is within the proximity of 55^0C or not.
- Whether the temperature at the inlet of the engine is within the proximity of 55^0C or not.

DRUPS Room CFD Case Study

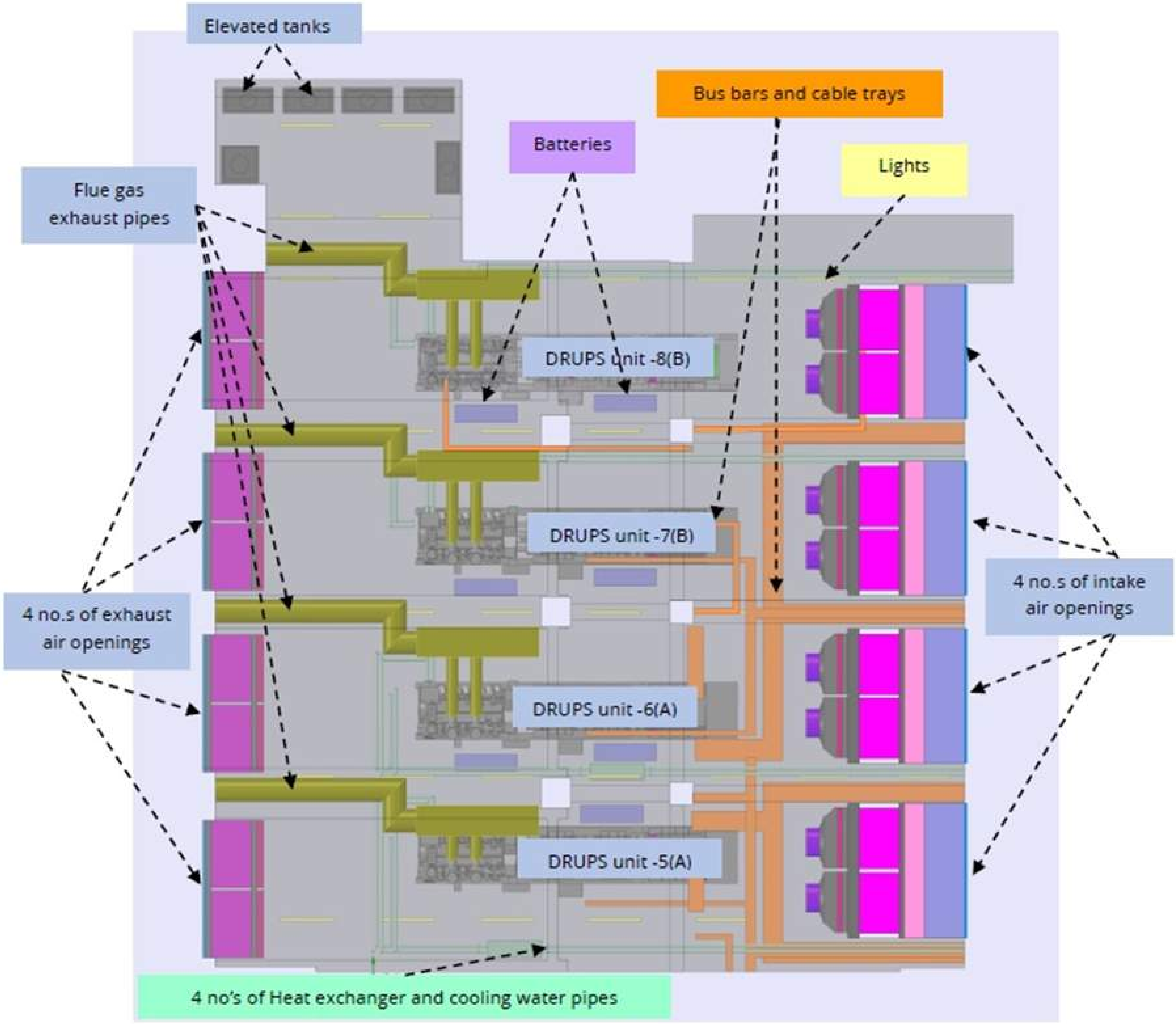


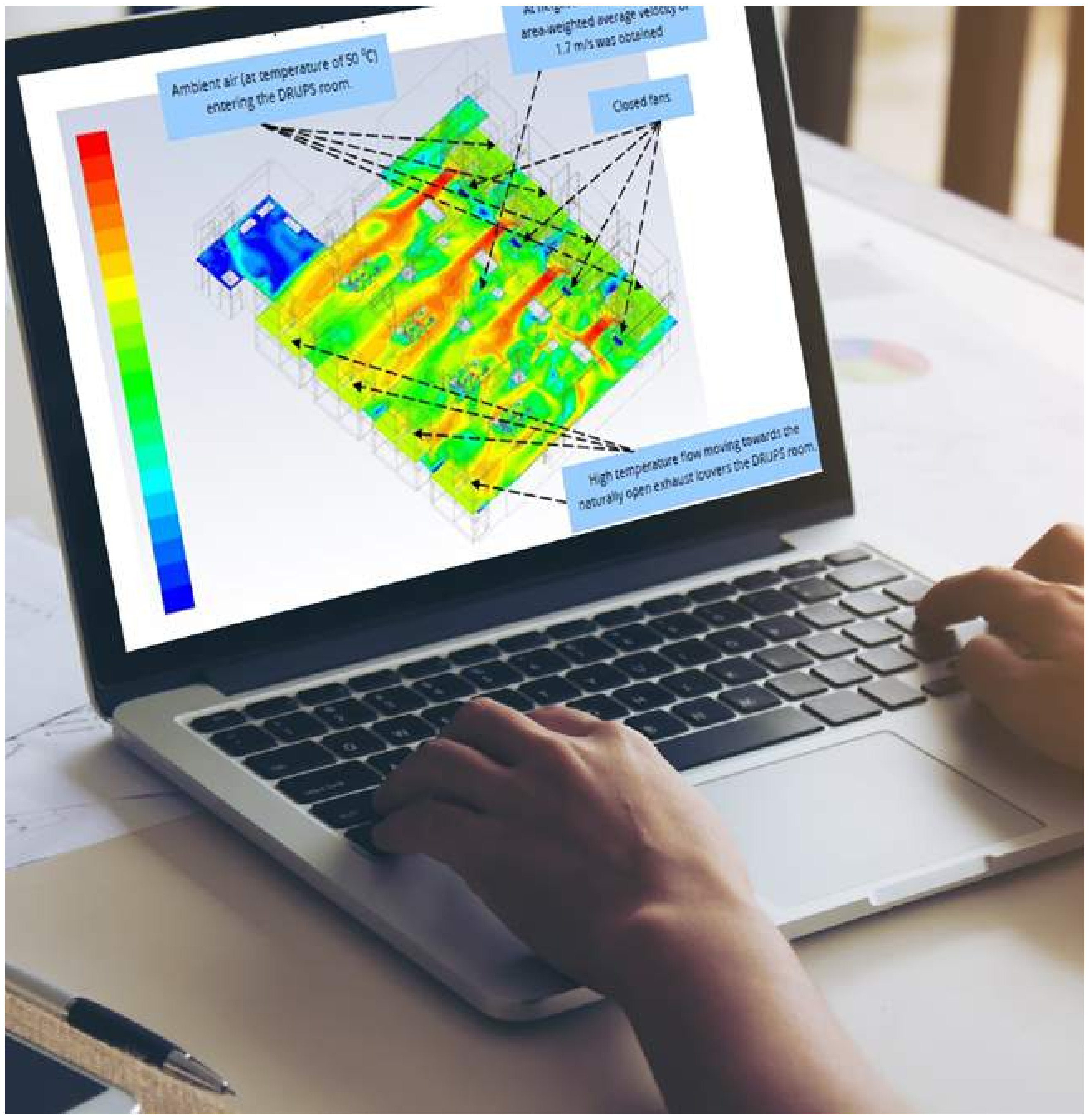
Fig. showing a top view of DRUPS Room located on a Ground floor of a Data Center.

CFD analysis was carried out using ANSYS CFD software.

Modeling & Analysis

The fluid flow (movement of air) is considered to be incompressible in the CFD model since there is no significant variation in the fluid density with respect to the time within a fluid parcel when it is in motion. To build a mathematical representation of a 3D model and numerically solve the 3D Navier-Stokes equations over a discretized flow field, the k-w SST turbulence model is used for simulating the turbulence behavior of the flow.

The ventilation system design consists of 8 fans (with 7 fans in working & 1 fan in OFF mode) for each DRUPS unit. Due to the suction effect of the fans, ambient air enters one side of the room and the hot air will be extracted naturally through the louver openings located on the opposite side of the room.



Heat Load Details

No.	Parameter	Value
1	Heat load from the Engine	90 KW
2	Heat load from the Accumulator 1	30 KW
3	Heat load from the Accumulator 2	30 KW
4	Heat load from the Alternator	84 KW
5	Surface temperature of generator exhaust flue pipes	60 ^{0}C
6	Surface temperature of other pipes	50 ^{0}C
7	Surface temperatures of the batteries	50 ^{0}C
8	Ambient air temperature	50 ^{0}C
9	Heat emitted from each heat exchanger (4 no's)	15 kW

DRUPS Room CFD Case Study

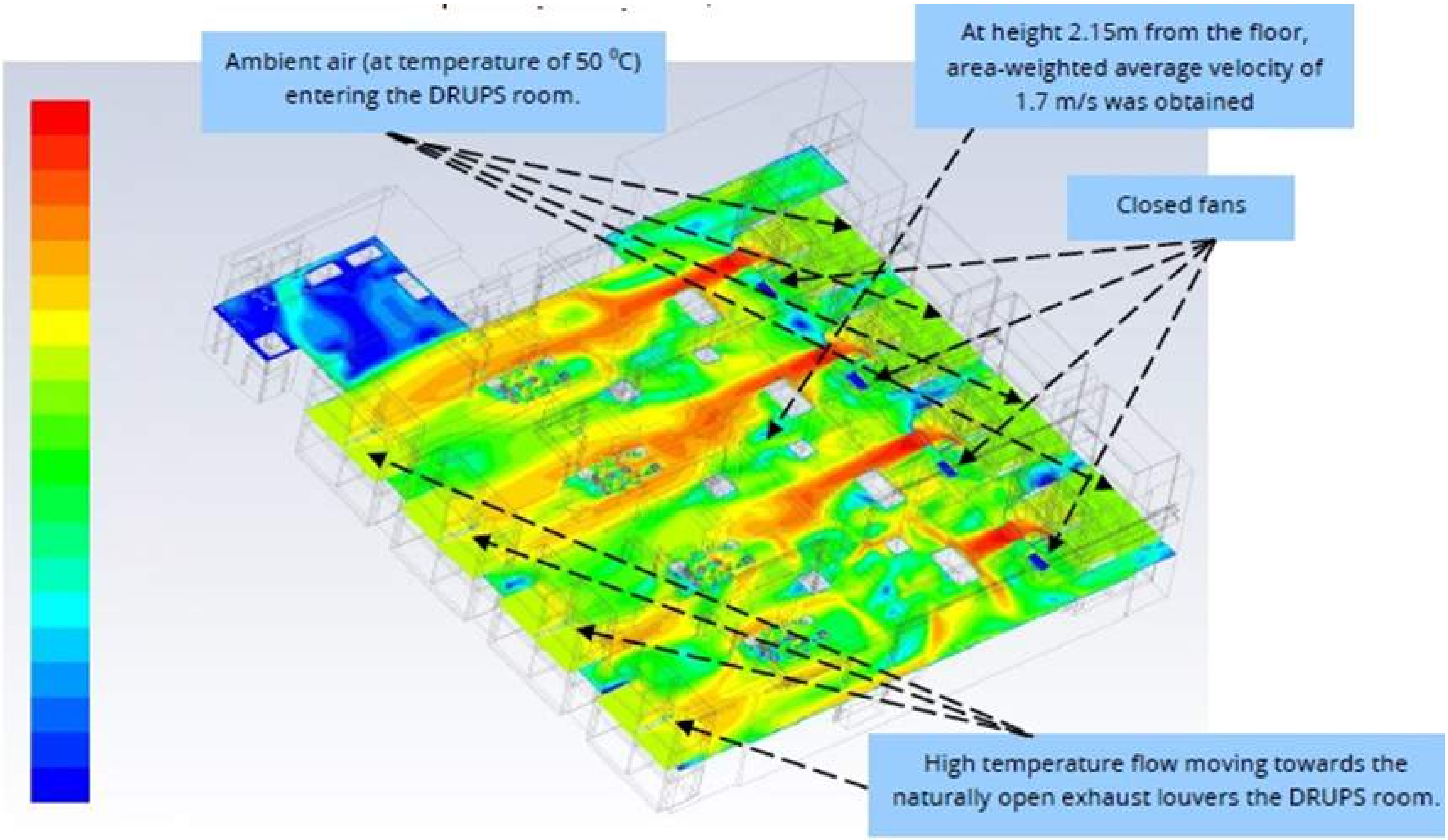


Fig. showing plot contour of velocity in m/s at 2.15 m ht.

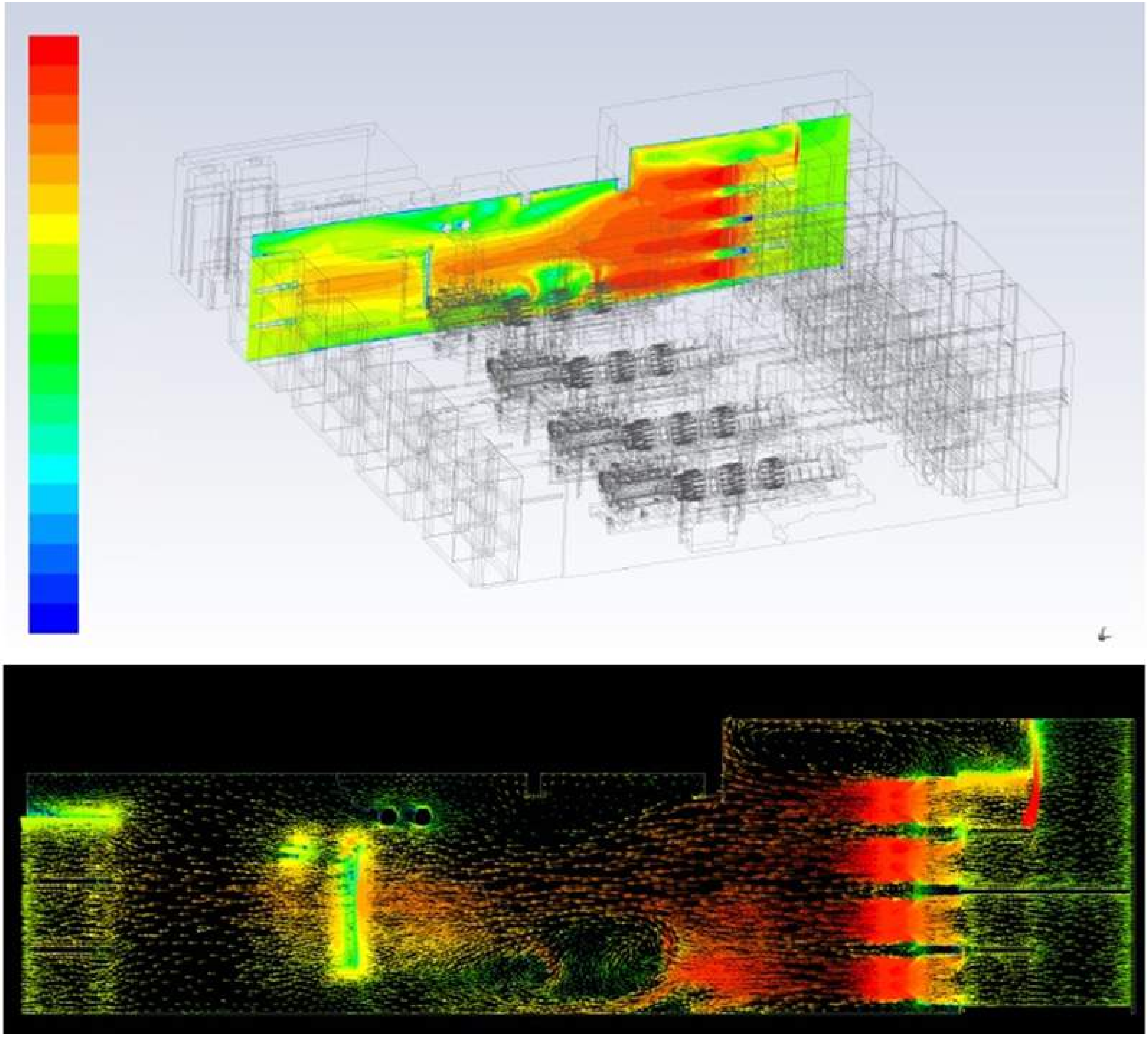


Fig. showing contour and vector plot of velocity in m/s in Y-plane in DRUPS Room

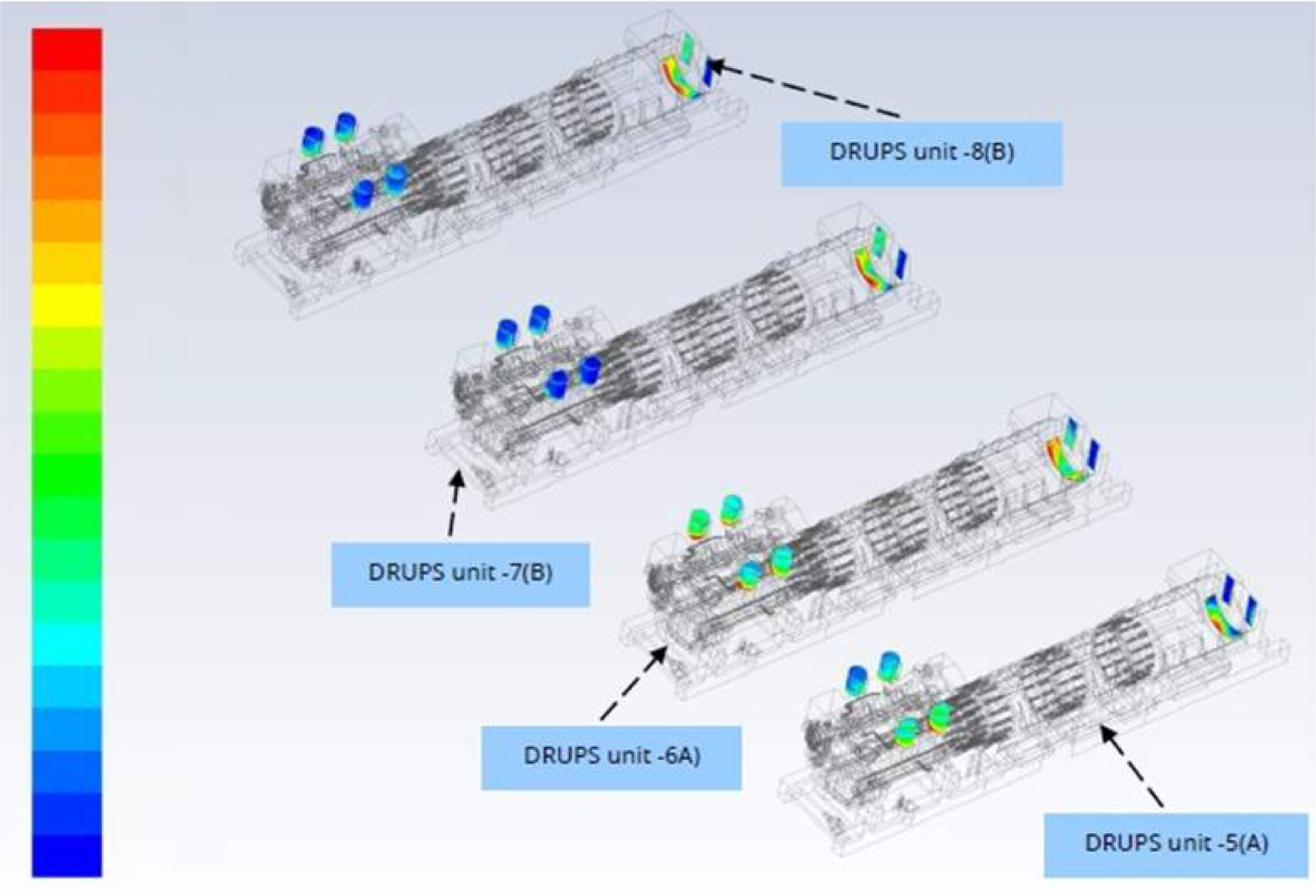


Fig. showing contour plot of temperature in 0C at alternator and engine inlet openings in DRUPS Room

Conclusion from CFD analysis

The average velocities obtained in the DRUPS room at various heights were in the range of 1.1 m/sec to 2.1 m/sec. Due to the position of the propeller fans, most of the ambient air from the fans is moving above the DRUPS units and flow inside the room is blocking the upward movement of the hot air (air heated up by the heat loads from the DRUPS units surfaces) and resulting the high temperatures nearer to the DRUPS units. This effect can be minimized by reducing the fan velocity or placing the fans at lower heights.

The average temperatures obtained inside the DRUPS room area at various heights was varying from 51.9°C to 61.1°C and in most of the areas around the DRUPs units, the temperature was more than 55°C.

Based on the CFD results observation, appropriate and feasible recommendations were provided to the client to improve the thermal and flow profile of the DRUPS room.

Mechartes is one such company with over 50 skilled engineers and simulation experts who work on root cause study. They have helped several clients globally in the past 17 years by analyzing existing plant/site conditions to curate the best-fitted design modifications and techniques for their clients.

The key benefit of CFD is it saves Time and Money as it is "Simulation-Based Design" instead of "Build and Test".

Services by Mechartes

Services by Mechartes

We at Mechatres are focussed on providing accurate simulation results with a professional and engineering approach. Our simulations represent the system closely and physics correctly at each parameter and step. With expertise in advance numerical tools like Computational Fluid Dynamics (CFD) and Finite Element Method (FEM), we offer following services in the Building sector:

- HVAC analysis of Generator yard
- AHUs, Chiller Yard & Cooling tower recirculation study
- Flue Dispersion Modeling
- CFD Thermal comfort analysis for indoor and outdoor environment.
- TES Tank & Buffer Tank Design
- Generator, UPS and DRUPS Room Analysis
- Smoke and evacuation modeling.
- Acoustic Modeling
- Piping Stress & Support design for HVAC/MEP and Pump rooms
- Seismic & Vibration Analysis
- Surge Analysis

You can find more case studies and other resources at our website.

Contact US

Global Sales Office

Mechartes International FZC
PO Box 125396, Dubai, UAE.

+971 4 381 2060

+971 56 1156701 +971 56 1156702

srihari@mechartes.com sales@mechartes.com

Global Engineering Office

Mechartes Researchers Pvt. Ltd.
D-57, Sector 6, NOIDA, Uttar Pradesh, India - 201301

+91 880 017 7554

+91 120 454 0208 +91 120 413 1889

contact@mechartes.com fahad@mechartes.com

That was...

CFD Modelling in Generator Room / DRUPS Room / Transformer Room

Mechartés
SIMULATION EXPERTS