Using Autodesk CFD in AEC Industry for Sustainable Design

Dr. Munirajulu. M

Chief Engineering Manager | @m_munirajulu

L&T Construction, Larsen & Toubro Limited, India

About the speaker

Dr. Munirajulu. M

24+ years industry experience in CFD analysis

Speaker at AU 2017, 2018, 2019 Las Vegas & AU2019 India

Using Autodesk CFD Simulation for MEP/AEC design

Focus areas: Data Center Cooling, Basement Car Park Ventilation, DG Room Ventilation, Smoke Simulation in Buildings, Air-Conditioning and Thermal Comfort Analysis

Sustainable Design in AEC

Sustainable design in AEC aims at using sustainable practices in design to reduce project's environmental footprint.

Sustainable practices in design include using digital technologies such as BIM (Building Information Modeling) and Simulation. This is all the more necessary due to off-site/modular construction and emphasis on use of high-efficiency heating, ventilation and air-conditioning systems. Off-site construction results in elimination of waste and use of high-efficiency systems result in reduced energy consumption. Digital design also leads to less rework and less wastage during construction. Simulation provides an opportunity to evaluate and analyze the performance of the design at concept stage and potential optimization at detailed design stage. All these contribute to sustainable design goals.

Autodesk CFD in AEC Industry

MECHANICAL VENTILATION

Applications include: Thermal comfort of occupants within a building, data center cooling and smoke extraction analysis for life safety.

NATURAL VENTILATION

Applications include: Wind driven ventilation and displacement ventilation/ thermal stratification

WIND LOADING AND RAIN WATER INGRESS

Applications include: **Wind flow over and around buildings for ventilation, wind loading on building facades and roof, and wind driven rain ingress into buildings**

FREE SURFACE FLOW/ RAIN WATER FLOW

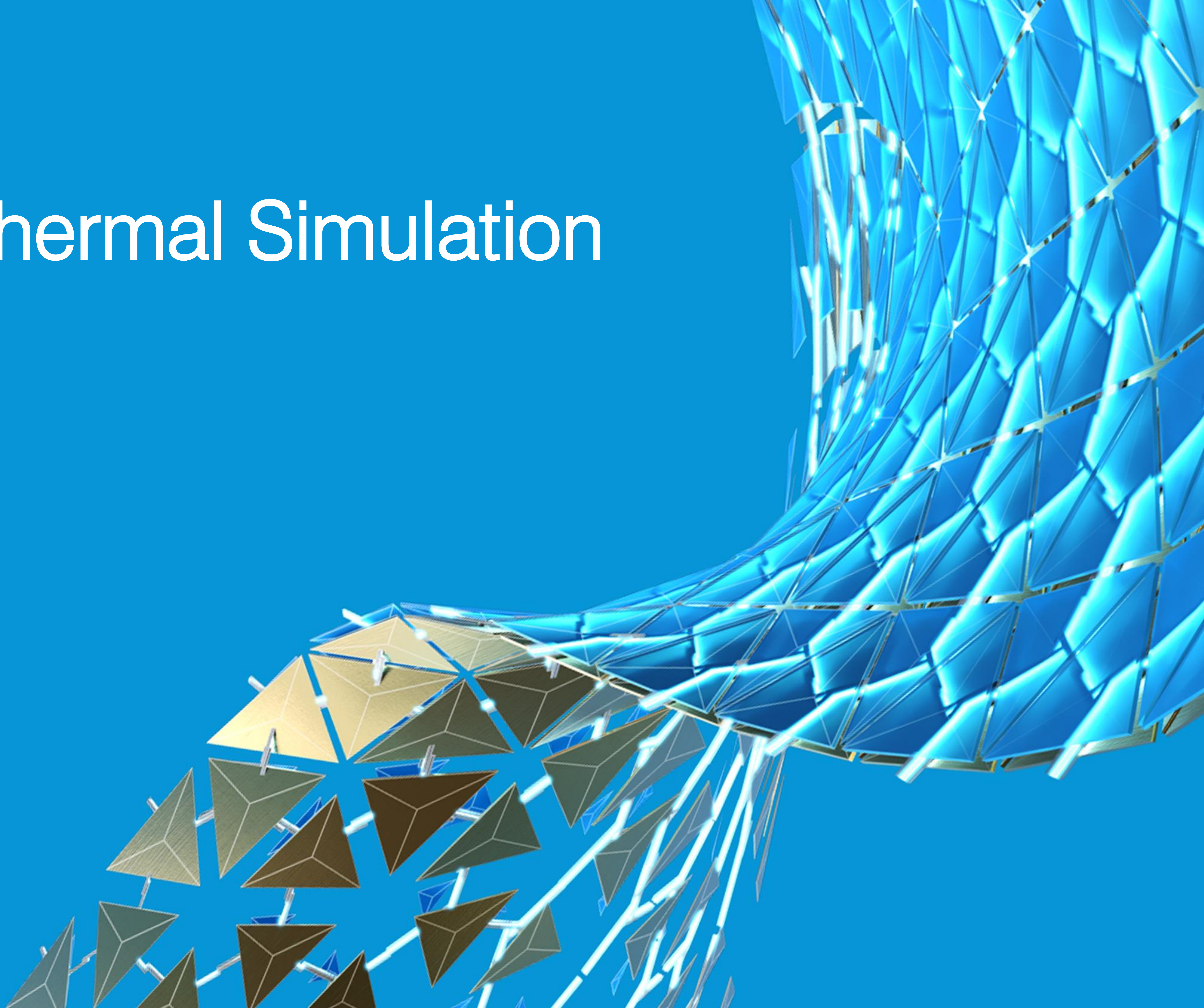
Applications include: **Rainwater flow over airport roof and roof drainage design**

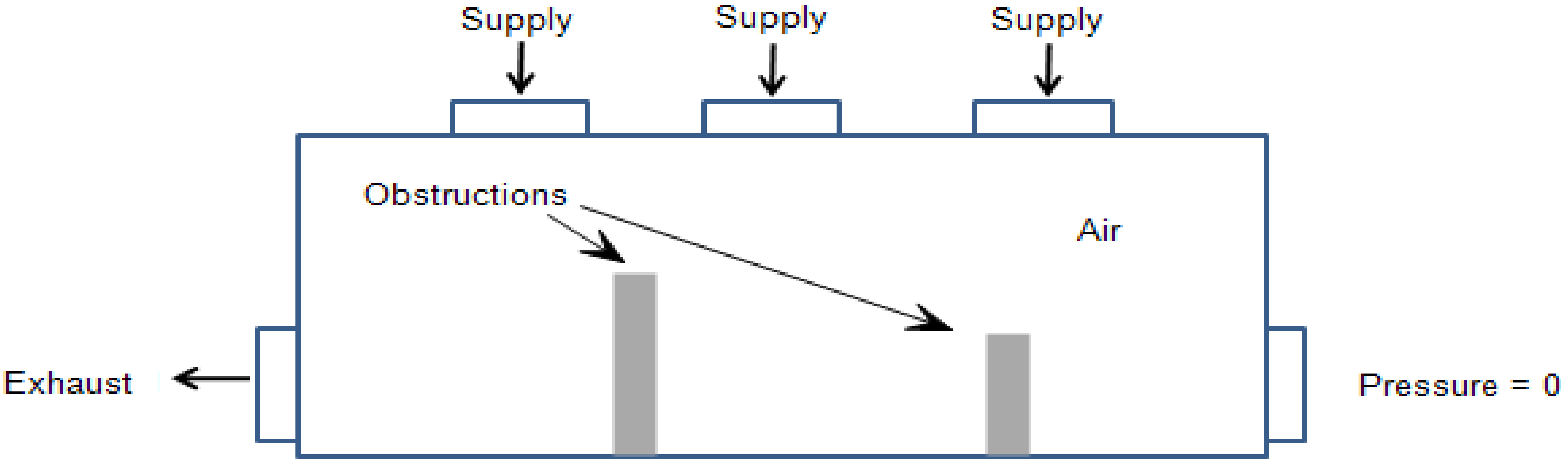
Key Learnings

Autodesk CFD is capable of simulating airflow and temperature distribution in AEC applications. Outcome of CFD simulation and analysis are helpful in performance-based design of buildings

- Air flow and thermal simulation capabilities in Autodesk CFD
- Modeling and simulation techniques for energy efficient ventilation design of occupied spaces
- Modeling and simulation of wind flow and rain flow on airport building
- Relevant results from CFD for sustainable design

Airflow and Thermal Simulation

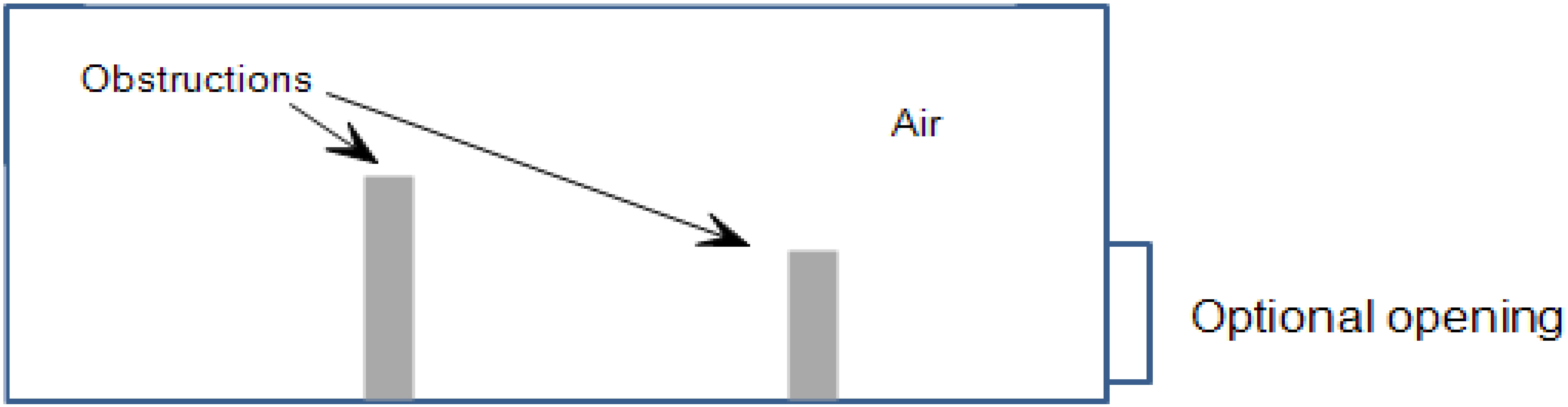




Mechanical Ventilation

Mechanical ventilation creates air movement, driven by fans and provides air change rates to remove stale air and replace it with ‘fresh air. This results in human comfort and good indoor air quality. In Autodesk CFD, mechanical ventilation analysis deals with: air regions, internal obstructions, supply vents/diffusers/fans, exhaust registers/vents/fans and thermal boundary conditions.

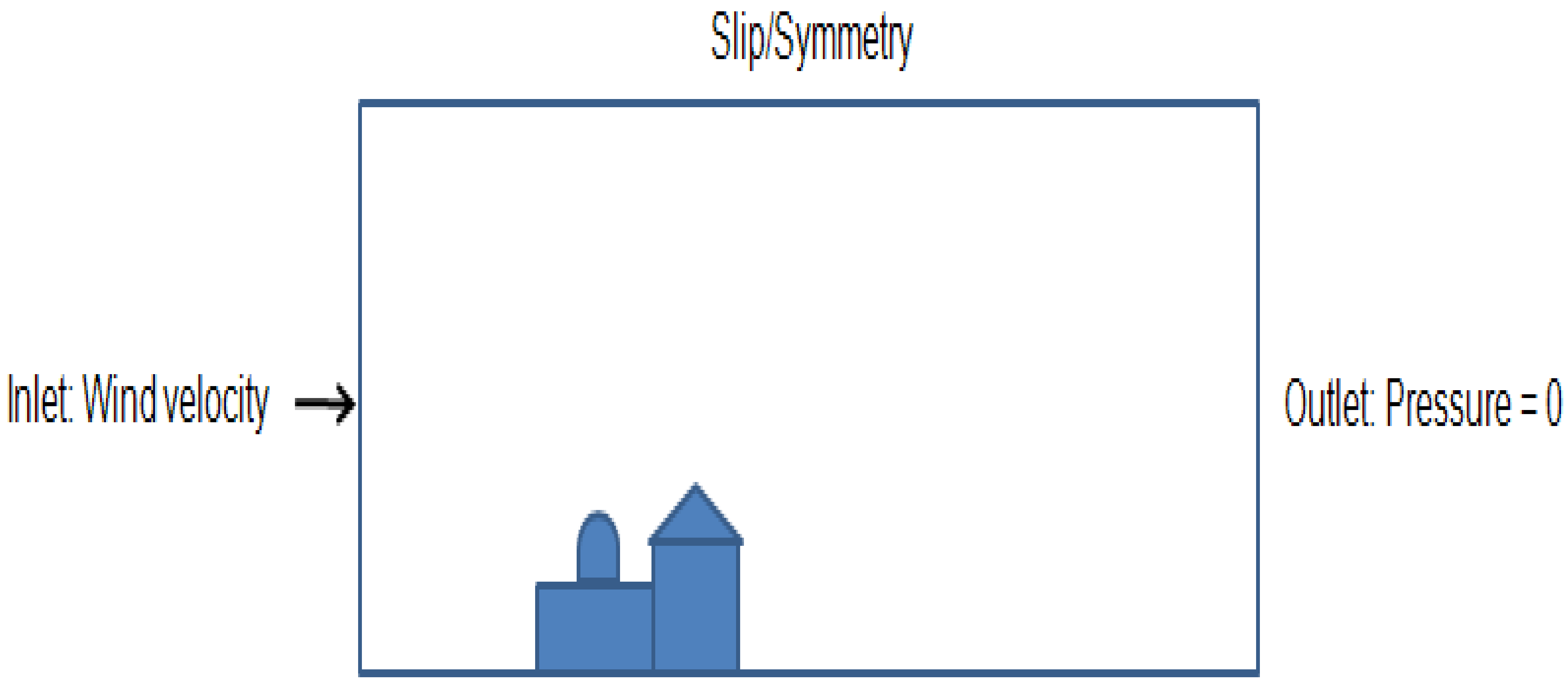
http://help.autodesk.com/view/SCDSE/2019/ENU/?guid=GUID-C52E55AA-38F7-4116-B83A-5AEDF9978F24

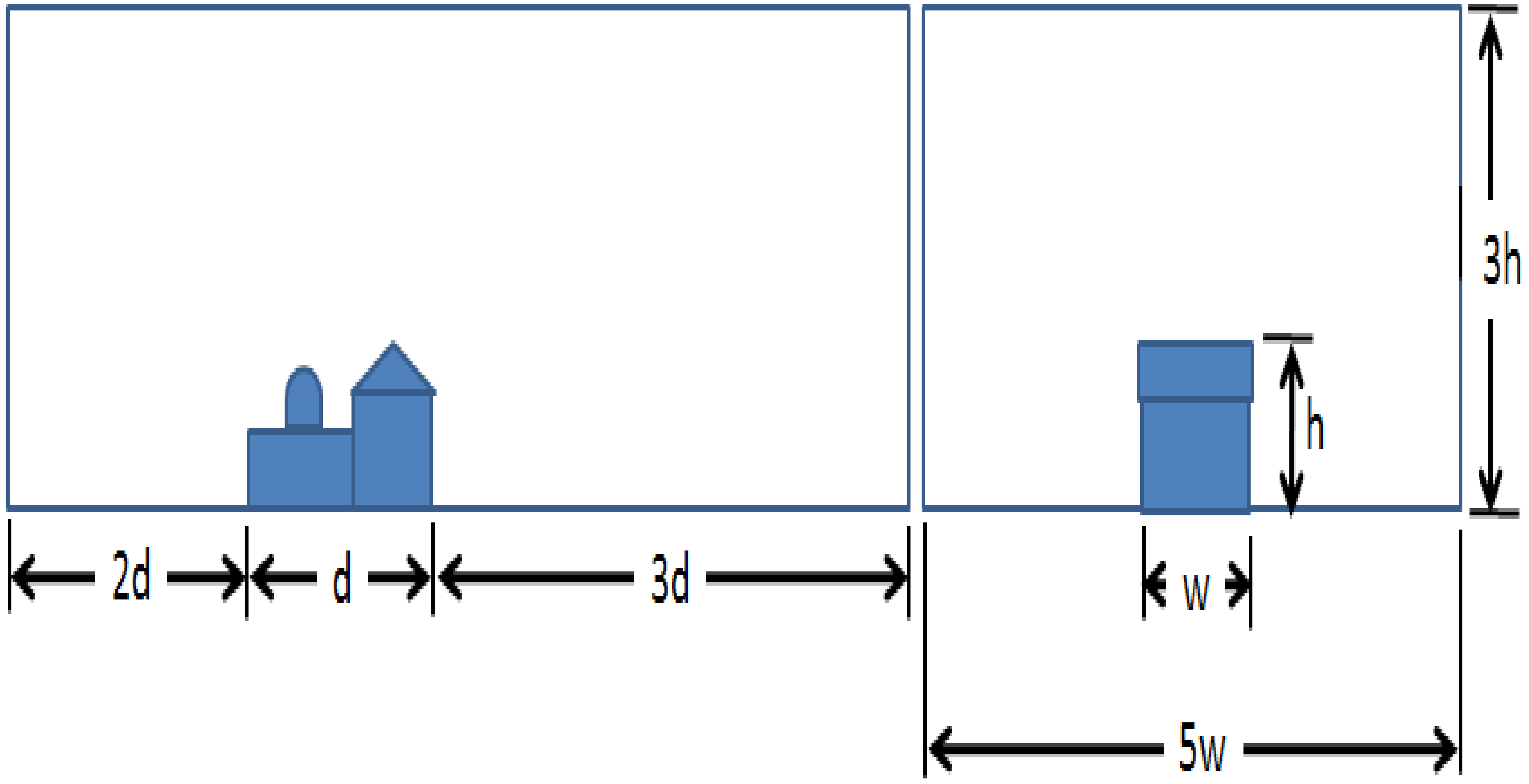


Natural Ventilation

Autodesk CFD can be used to simulate both wind driven as well as buoyancy driven air flow within buildings. Smoke extract analysis using smoke vents is an example buoyancy driven flow. Air movement within a building due to wind is an example of wind driven ventilation.

http://help.autodesk.com/view/SCDSE/2019/ENU/?guid=GUID-F6011744-B69B-47D0-BB2B-50E1BA8DC32D





External flow/ Wind loading

wind speed as a velocity boundary condition at inlet

Static Gage Pressure = 0 at outlet

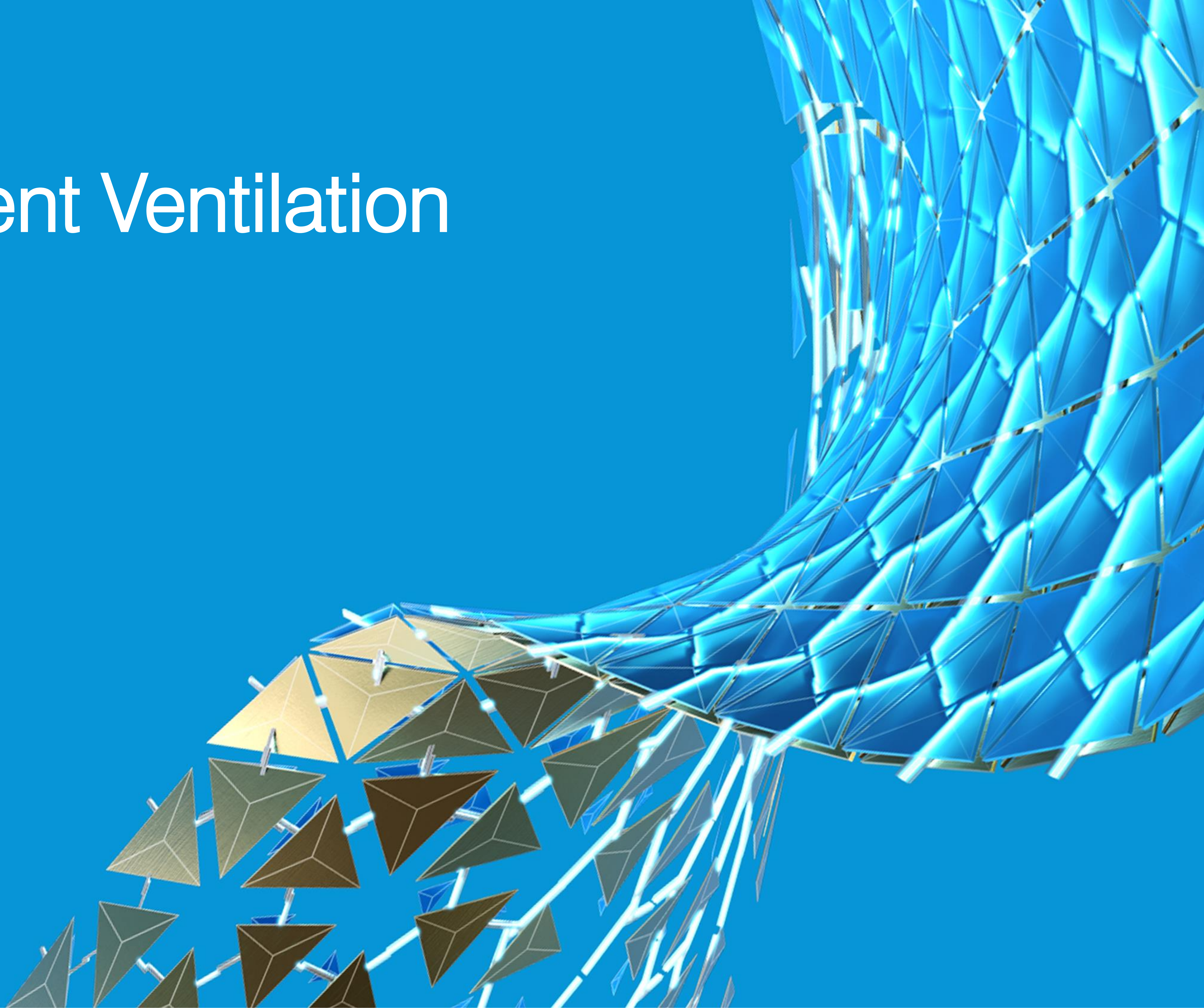
Slip/Symmetry to the top and sides of the region.

External environment

Recommended relative dimensions for external flow analysis in Autodesk CFD

http://help.autodesk.com/view/SCDSE/2019/ENU/?guid=GUID-2BF3C8A5-6D70-4F6A-A792-95A27C605E3D

Energy Efficient Ventilation



Thermal Comfort

Modeling and simulation strategies for energy efficient design of a occupied spaces cover 2 project cases

i. A large stadium sports facility
   - Energy efficient fans
     - Mechanical ventilation

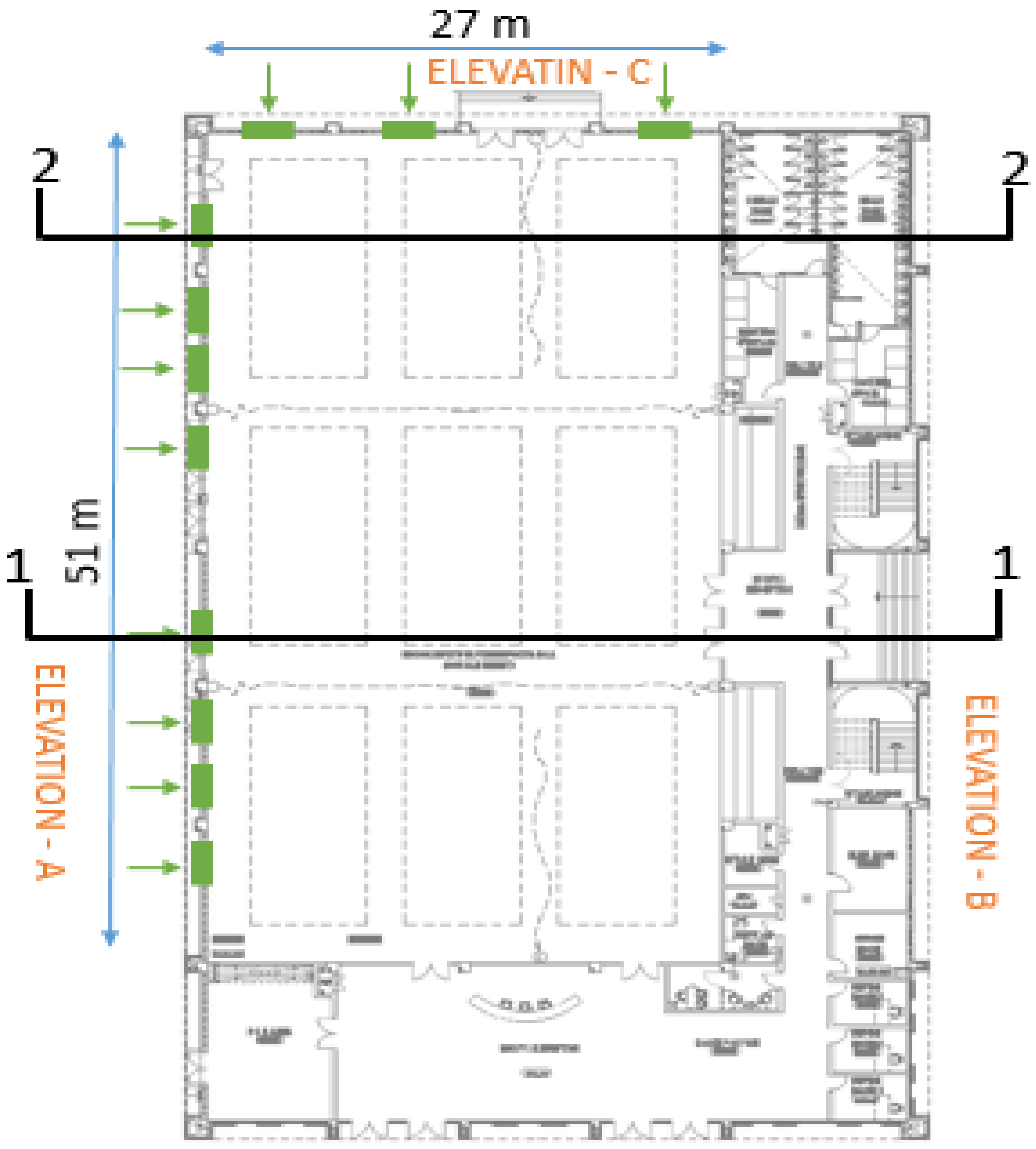
ii. A large convention center foyer
   - Displacement ventilation for thermal comfort
     - Buoyancy driven flow and heat transfer

(i) Sports facility- Design Scenarios

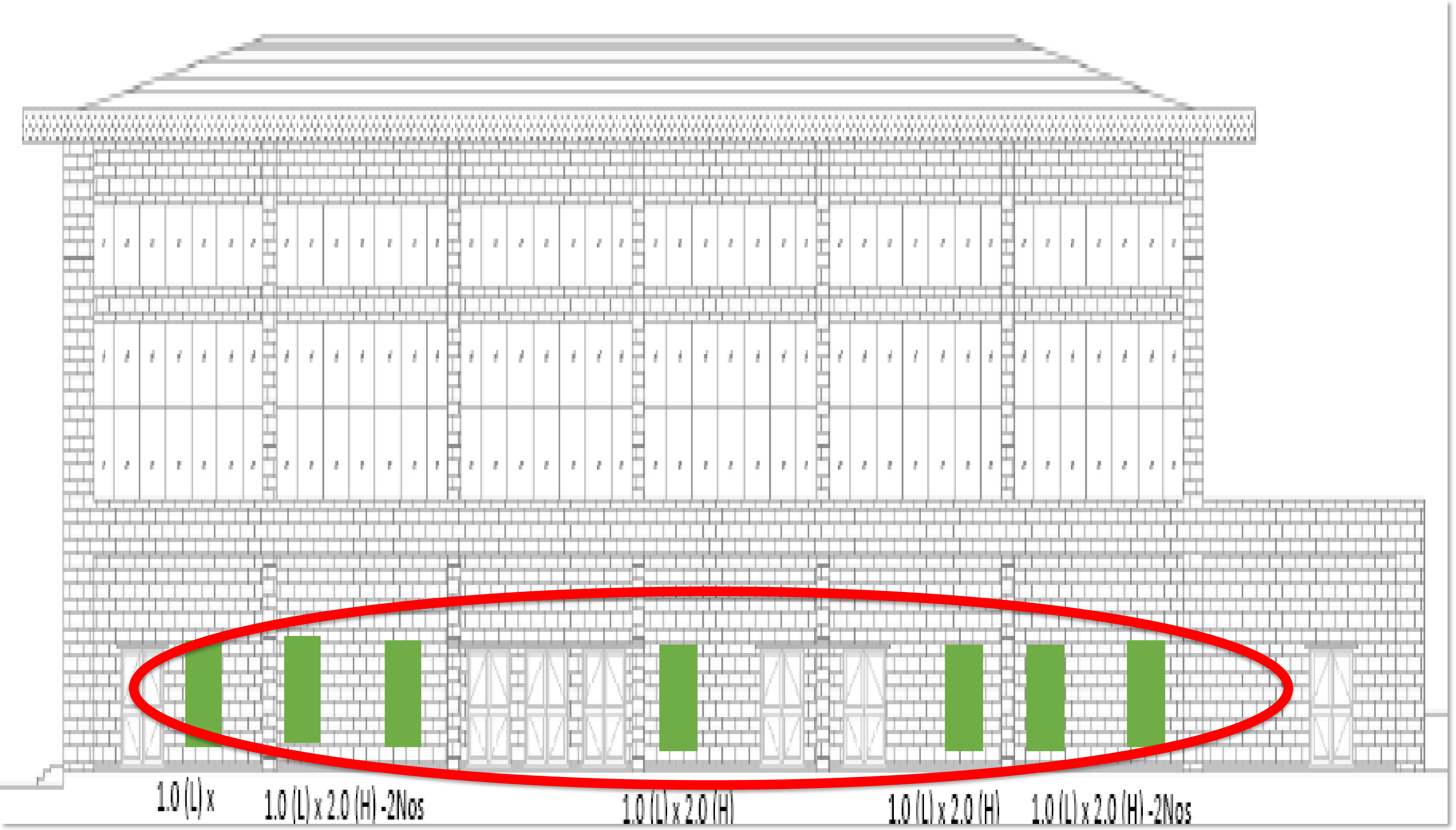
Autodesk CFD simulation is used to evaluate energy efficient fans for Sports training facility- multi-purpose hall. Following design scenarios are considered:

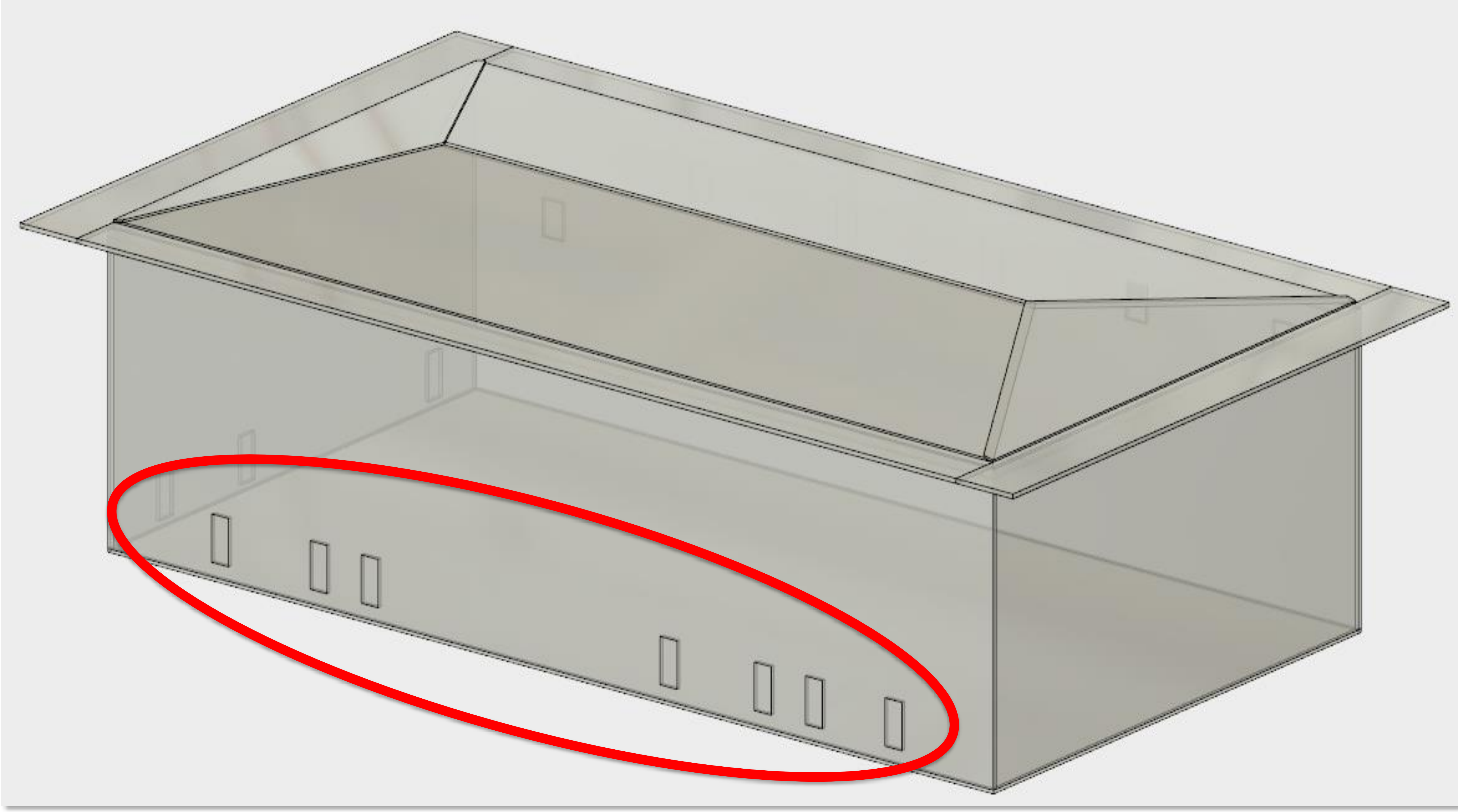
- Scenario 1: Axial fan scheme
- Scenario 2: Propeller fan scheme
- Scenario 3: Optimized propeller fan scheme

- Acceptance criterion for ventilation: Airflow velocities <0.15m/s in the playing zone at 4m level from the floor



Ground Floor



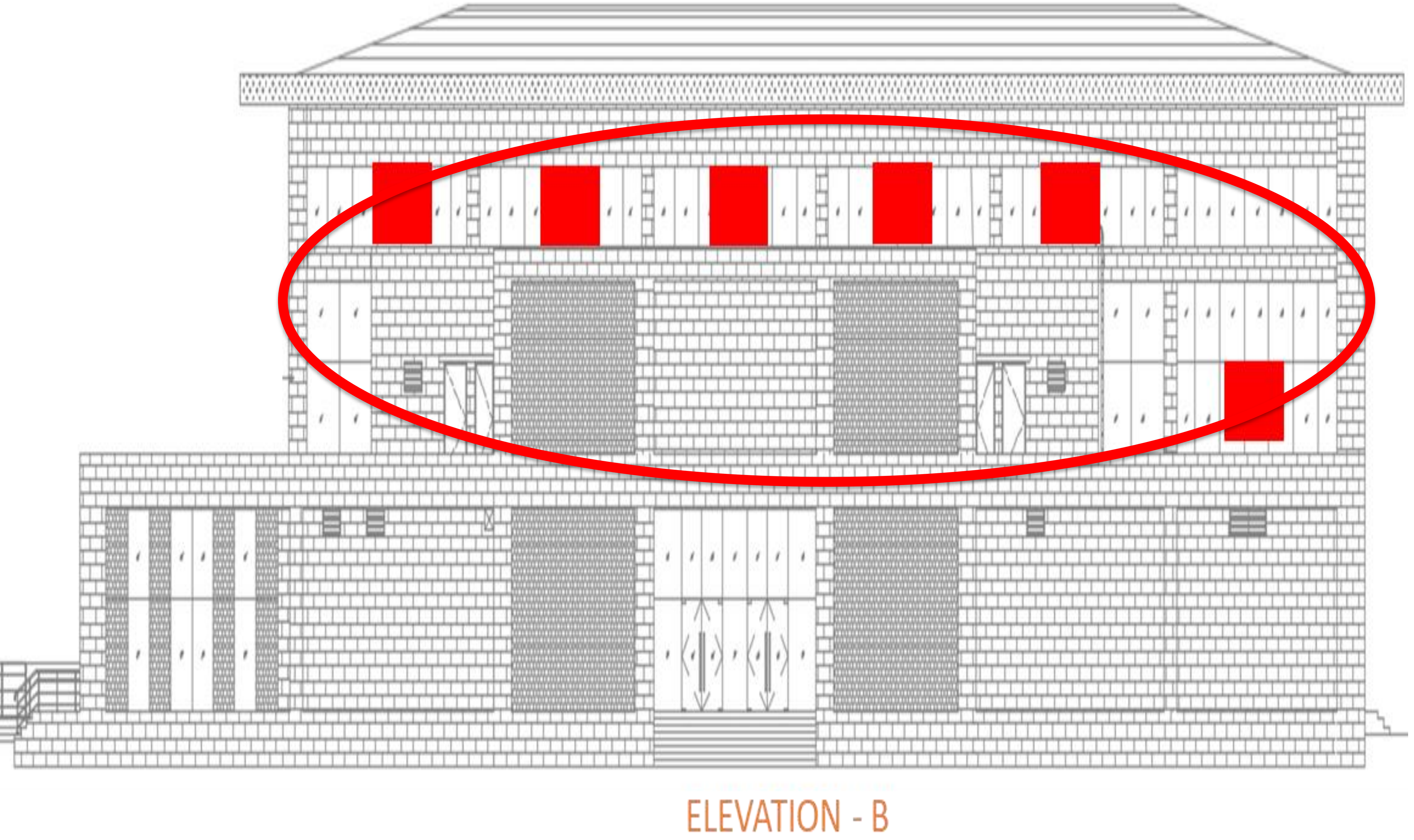


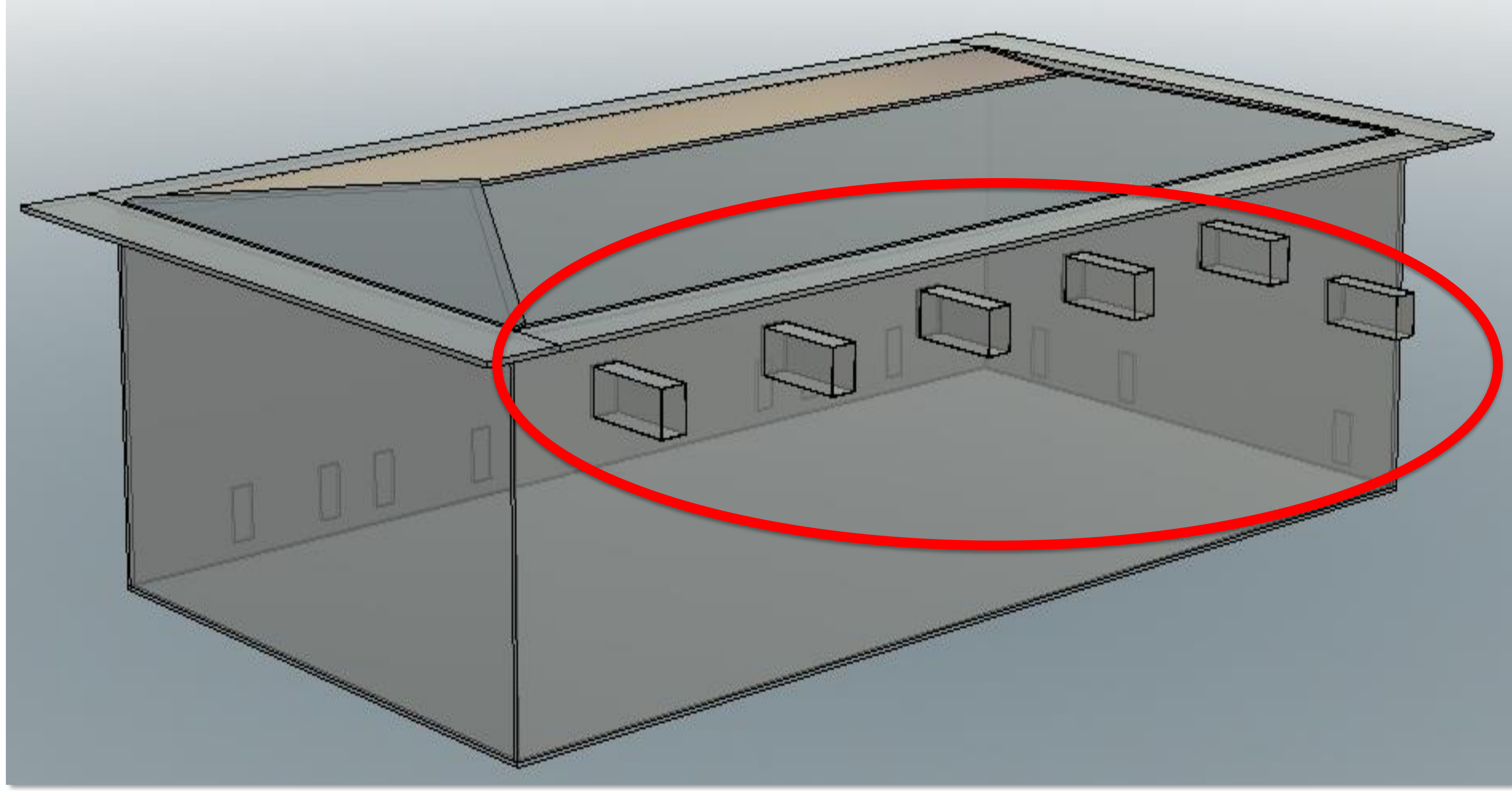
CAD model– Scenario 1

Ventilation air is drawn through louvers (circled)

CFD model – Scenario 1

Ventilation air inlet louvers in CFD model





CAD model– Scenario 1

Ventilation air movement is driven by axial fans near roof (circled)

CFD model – Scenario 1

Ventilation air outlets in CFD model

Scenario 1- Materials

Fluid volume is assigned “ Air” material with fixed properties

Inlet louvers assigned “Resistance” material with free area ratio

Scenario 1- Inlet extension and BC

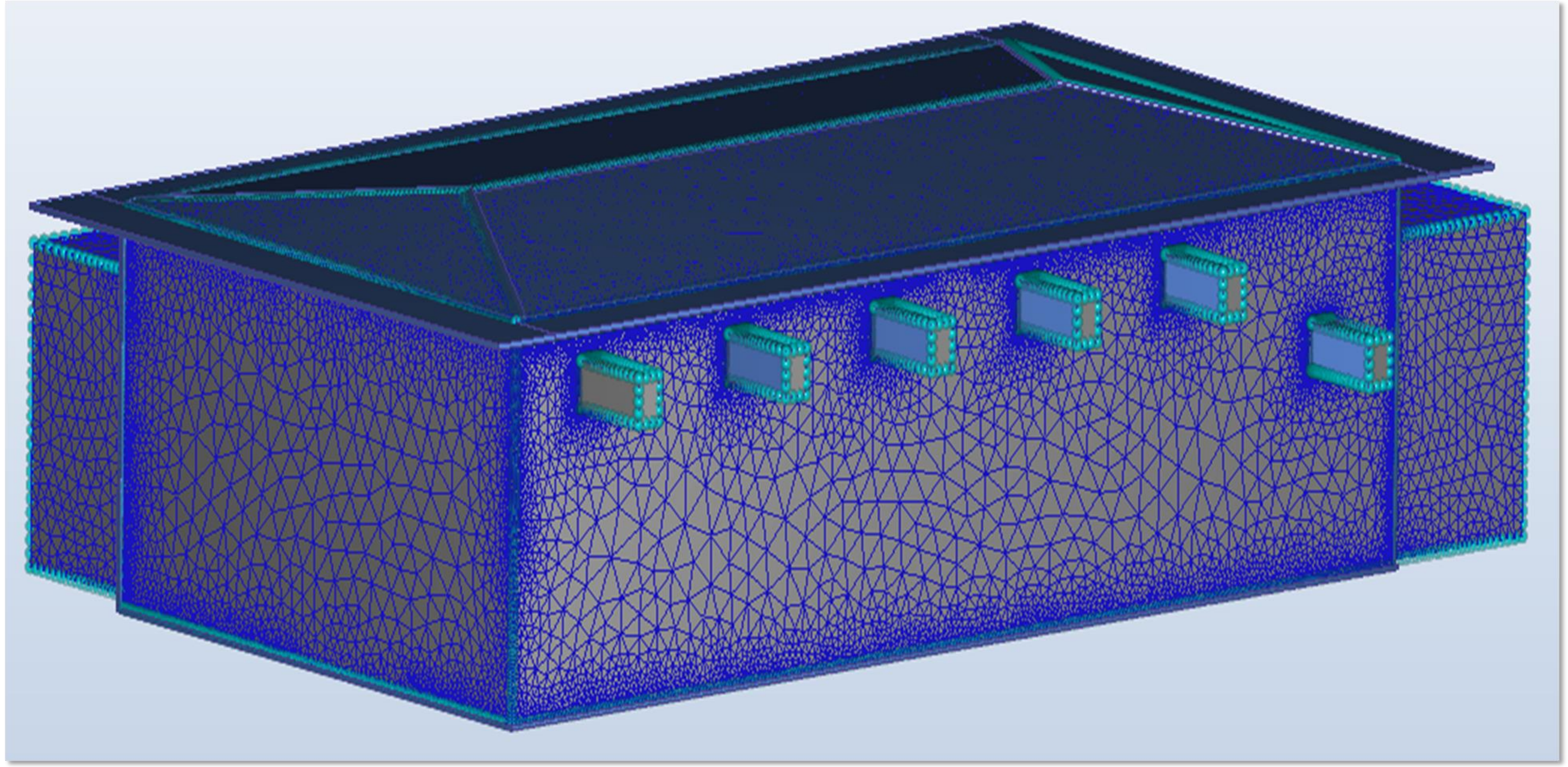
Inlets are extended by at least 1 hydraulic diameter

Inlet surface of external air volume assigned gauge pressure =0

Scenario 1- Outlet BC

Outlets are assigned volume flow rate BC for axial fans

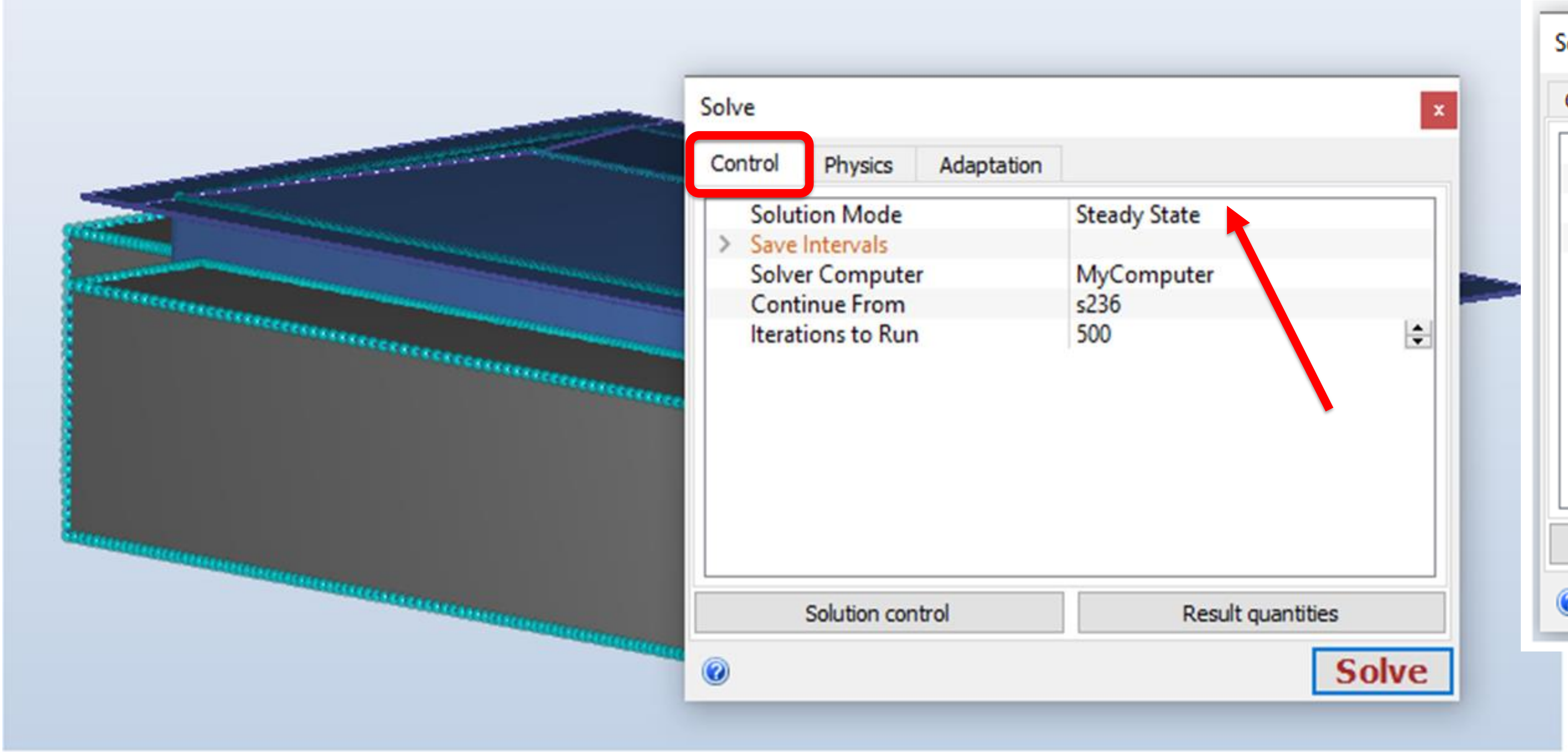
Direction of volume flow rate BC is set outwards to capture flow out

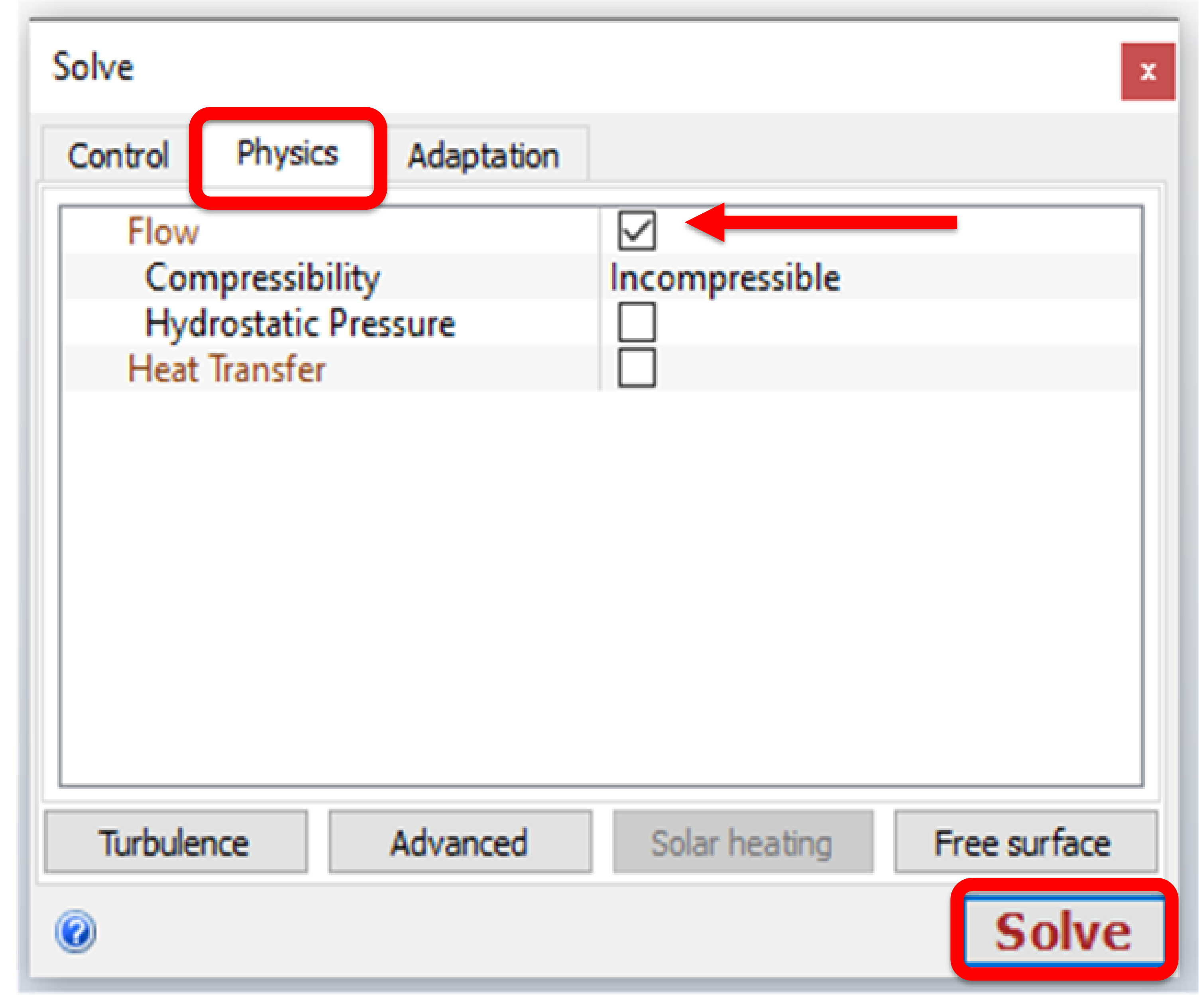


Scenario 1- Mesh

Automatic mesh

Inlet louver – refined mesh to capture flow

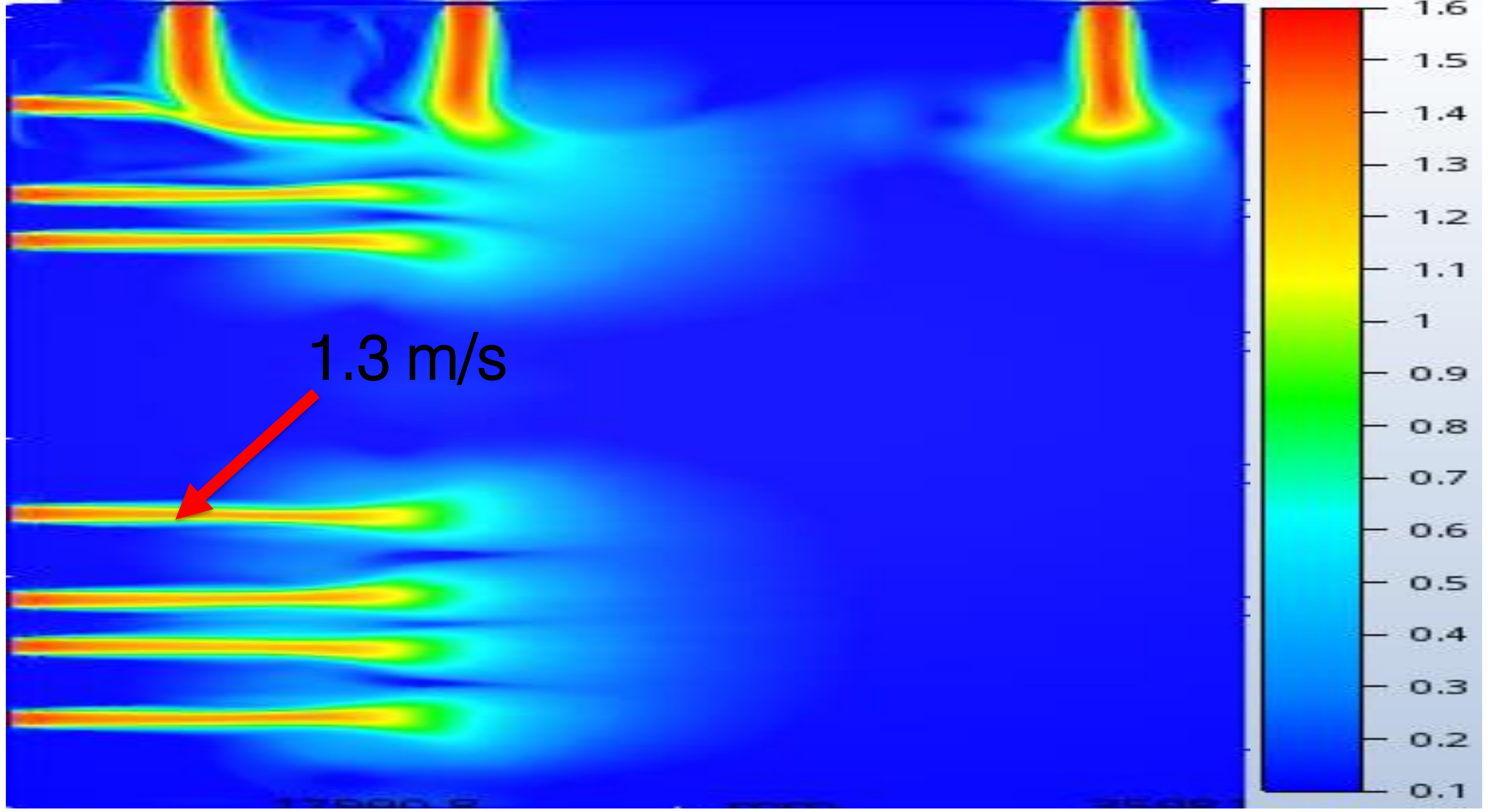


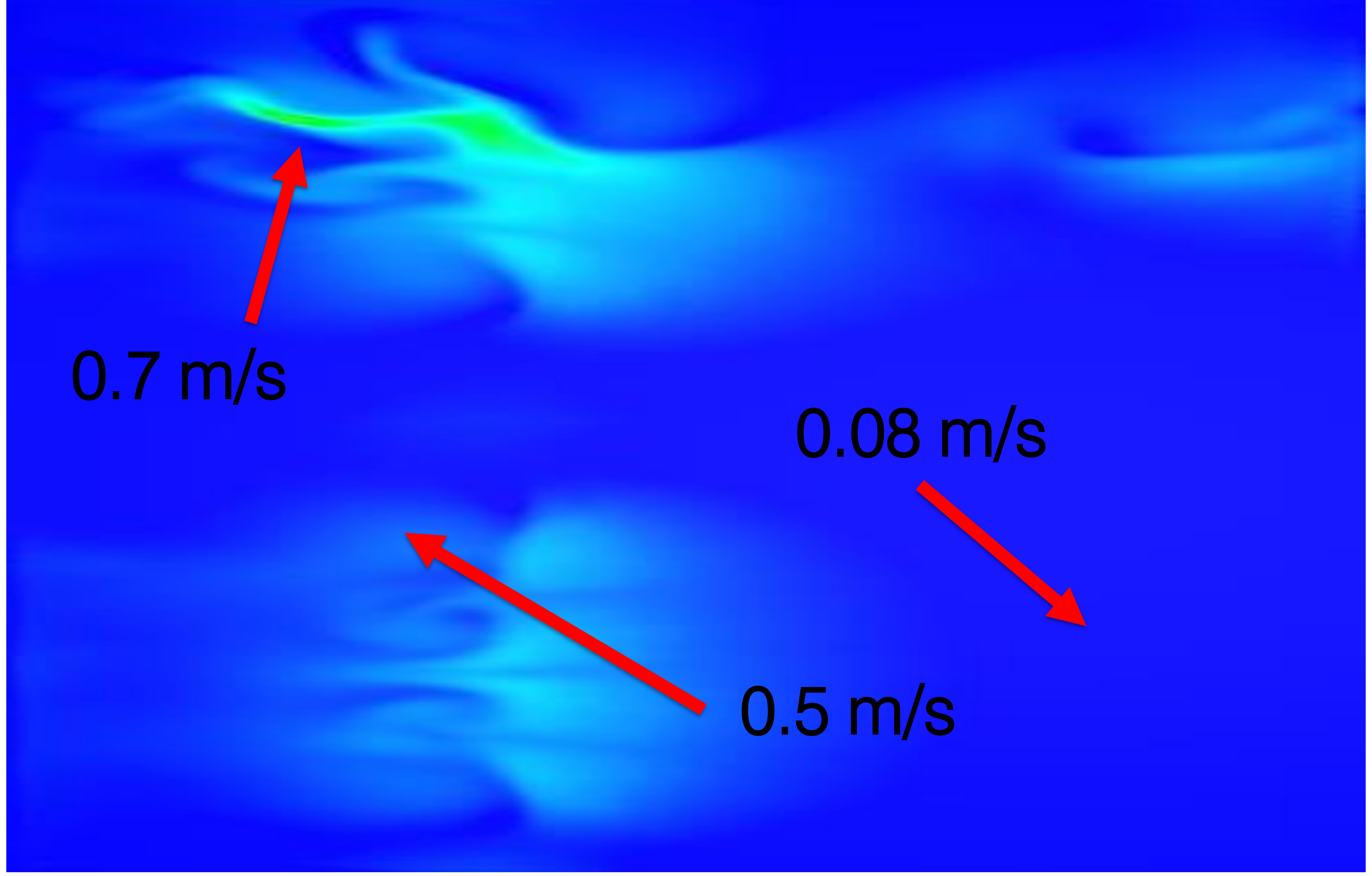


Scenario 1- Solver settings

Steady state solution

Flow = ON



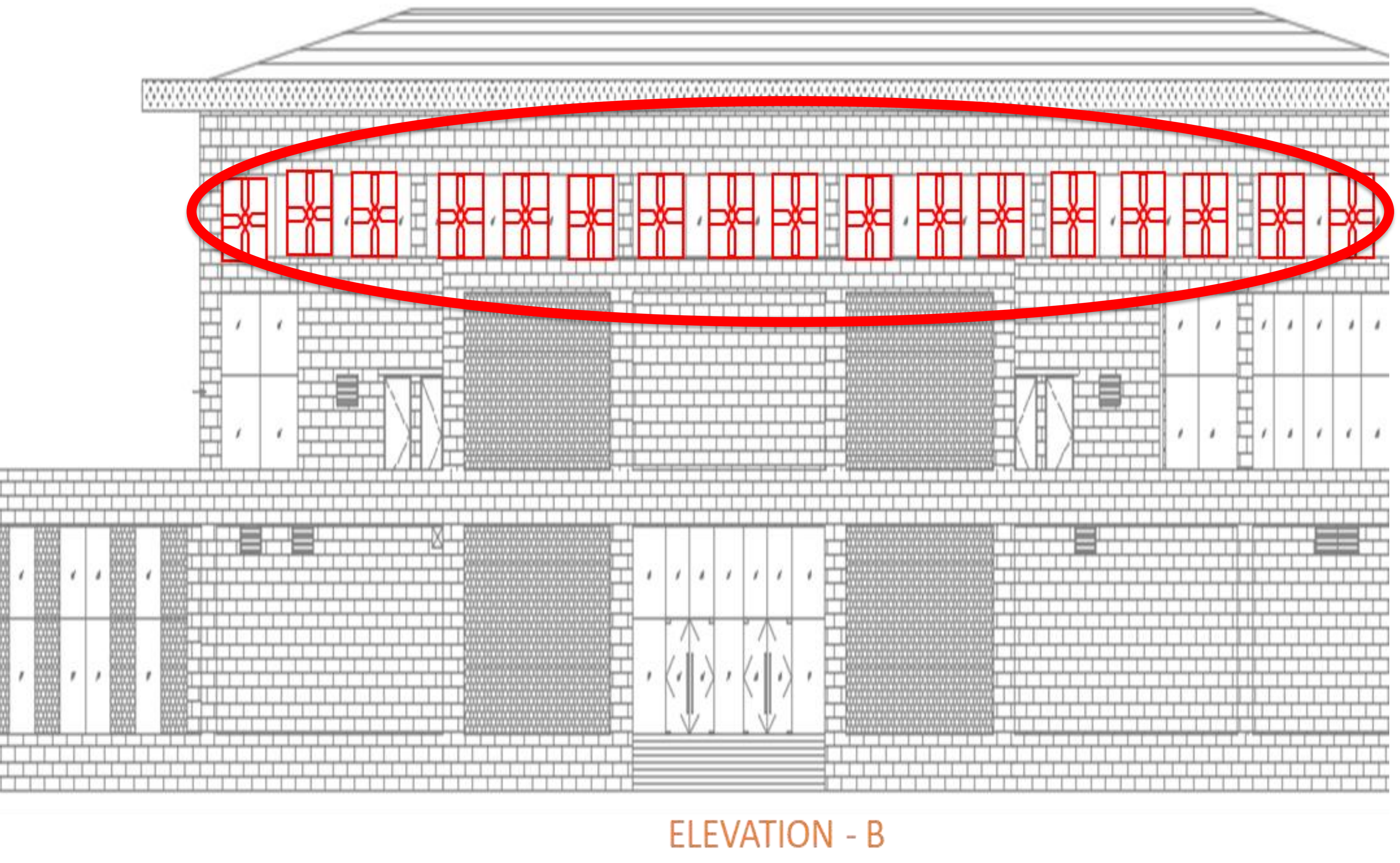


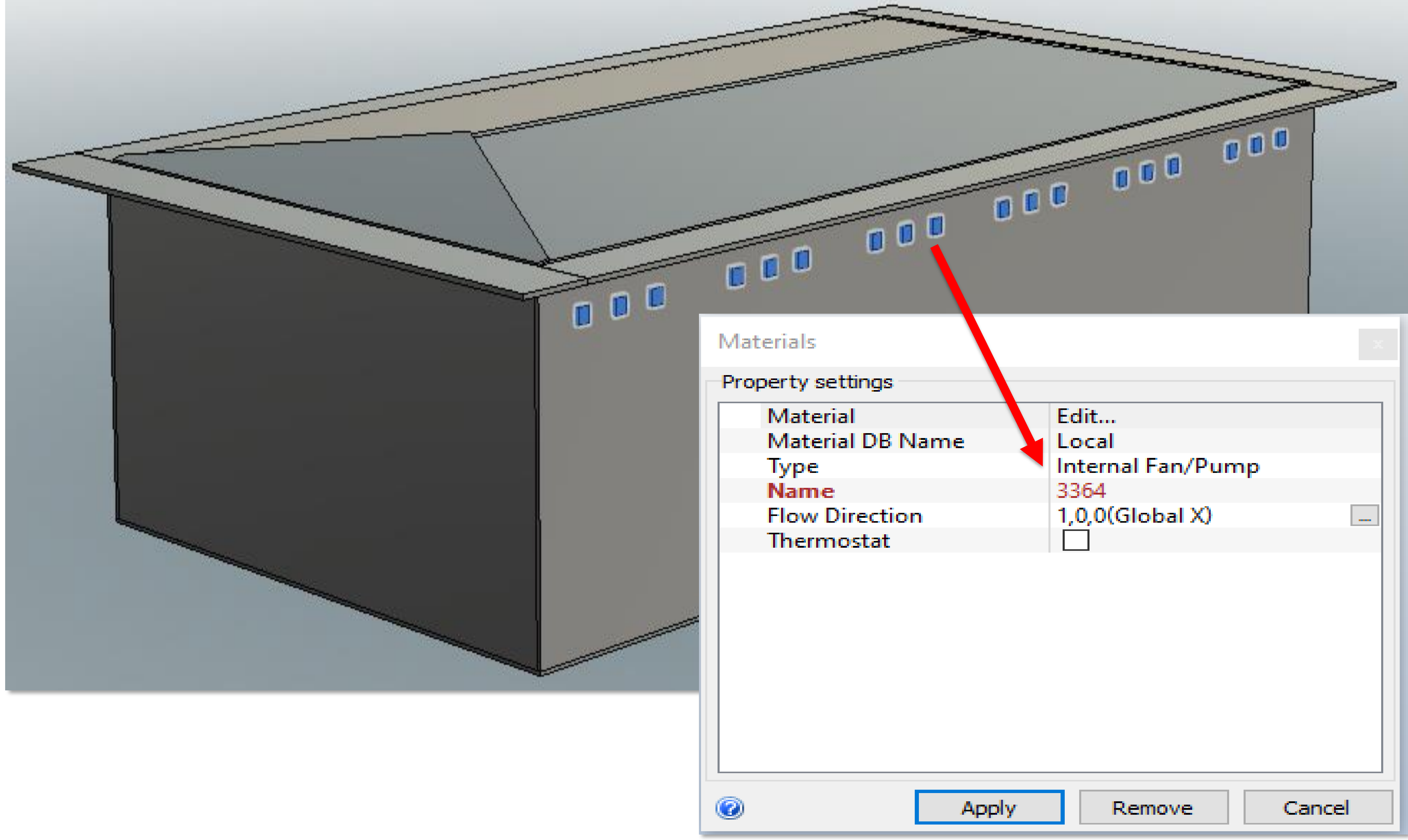
Scenario 1- Velocity profile at 1m

Streams of high velocity airflow coming out from the louver region into playing

Scenario 1– Velocity profile at 4m

Airflow velocity values are higher than 0.15 m/s





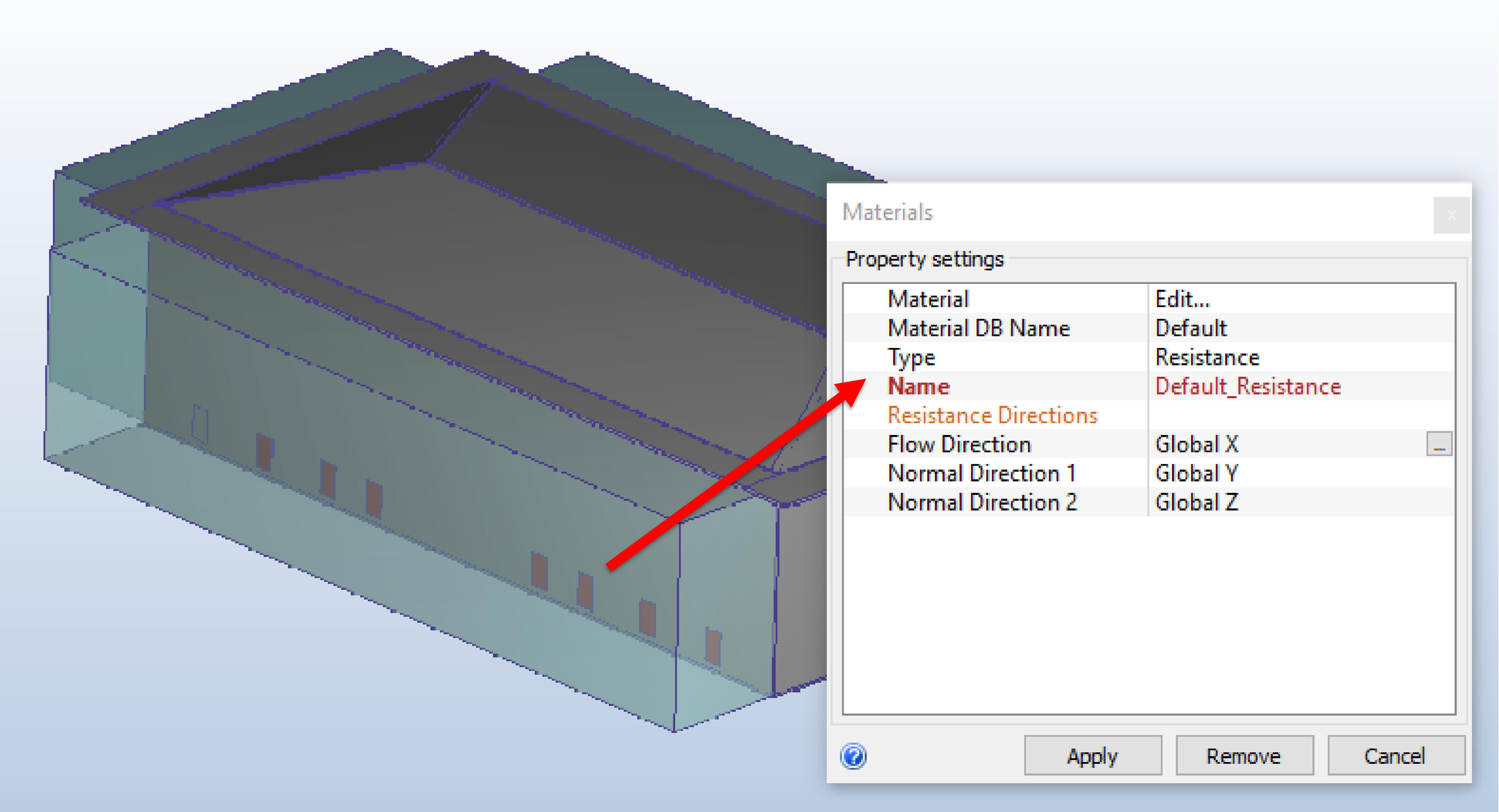
CAD model– Scenario 2

Ventilation air is drawn through louvers as in Scenario 1

Ventilation air movement is driven by propeller fans (circled)

CFD model – Scenario 2

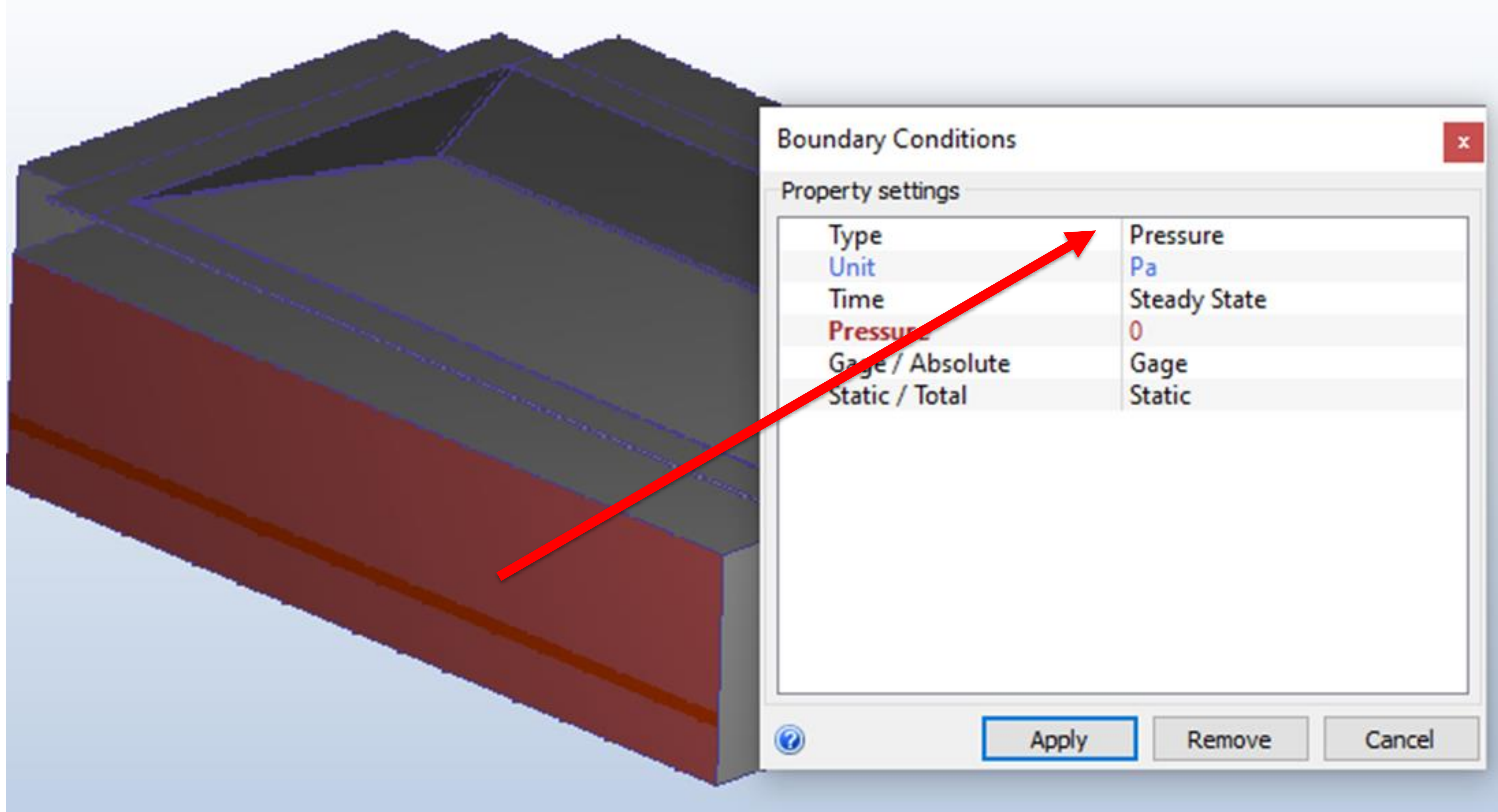
Ventilation air outlets modeled as "*Internal Fan* " in CFD



Scenario 2- Materials-Louver

Fluid volume is assigned " *air* " material with fixed properties

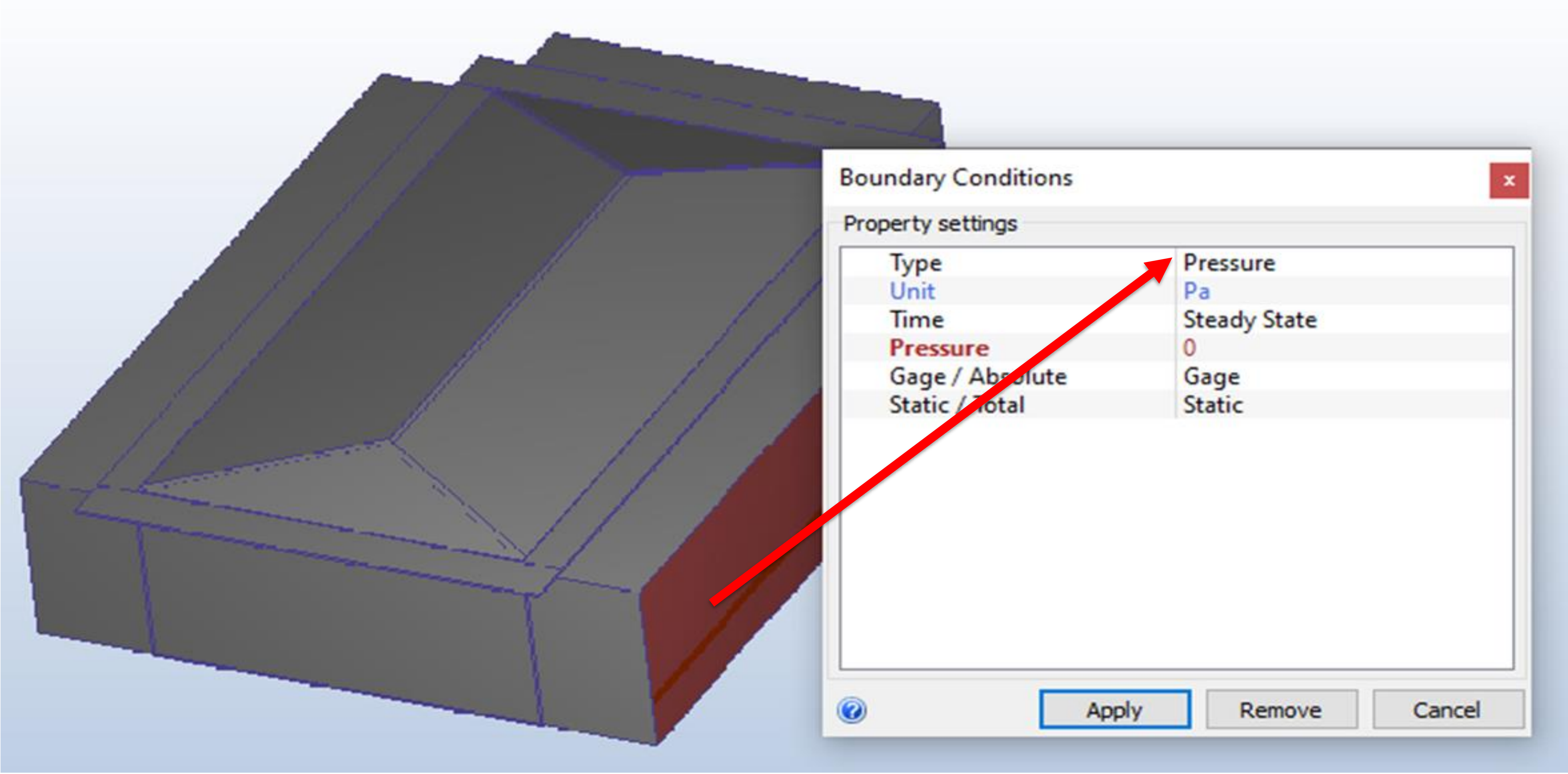
Inlet louvers assigned " *Resistance* " material with free area ratio



Scenario 2– Inlet extension and BC

Inlet is extended by at least 1 hydraulic diameter

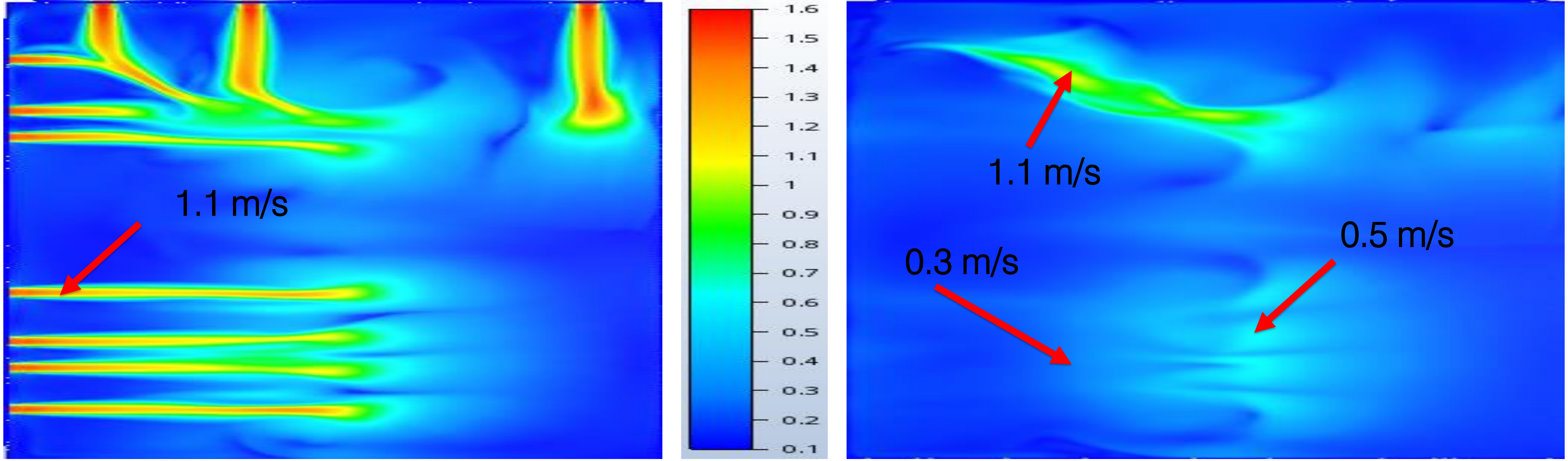
Inlet surface of external air volume assigned static gauge pressure =0



Scenario 2- Outlet extension and BC

Outlet surface of extended air volume is assigned "*Static gage pressure =0*" as BC

Propeller fans are modelled as CAD part and assigned internal fan material with CFM outwards

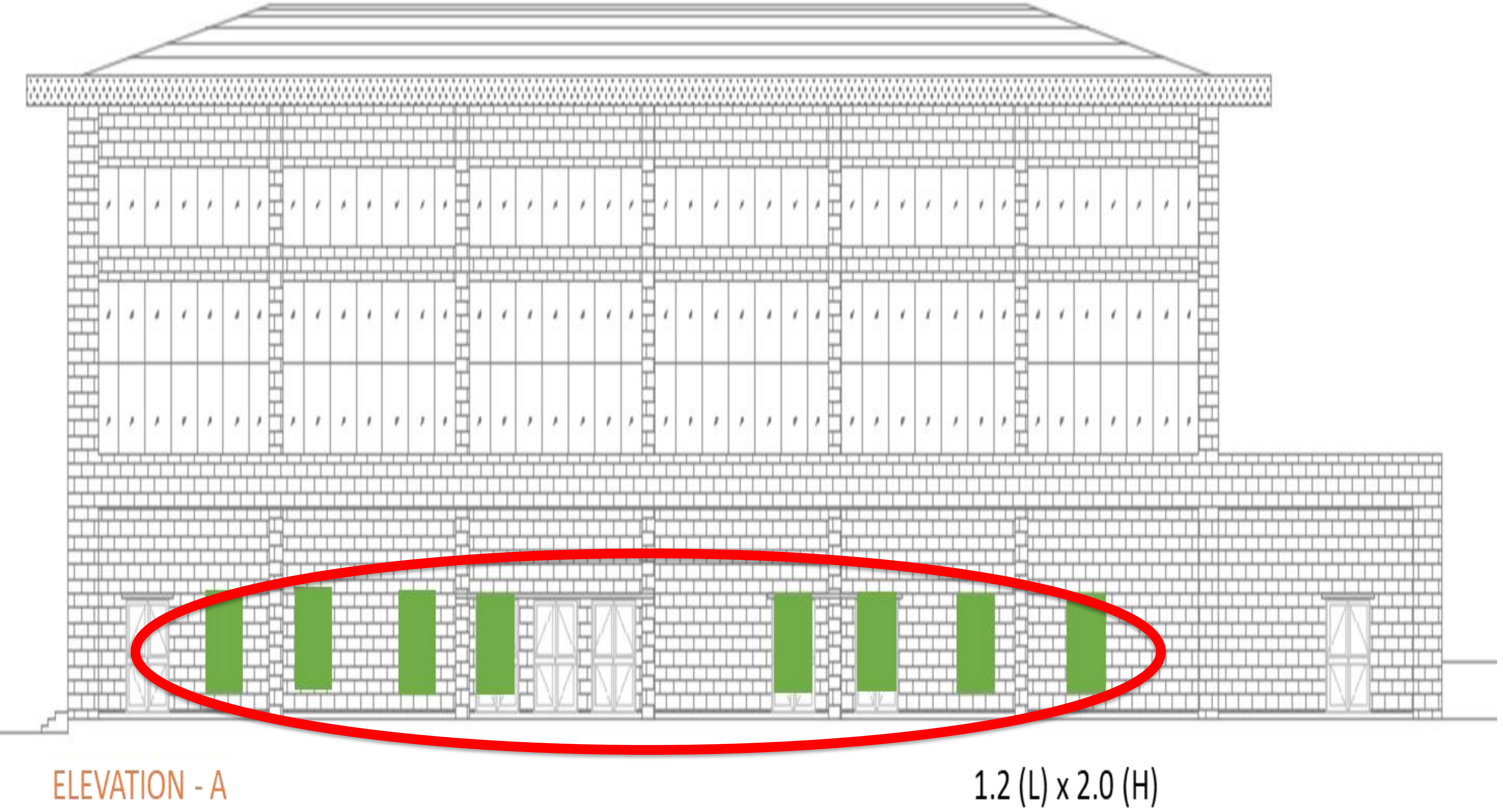


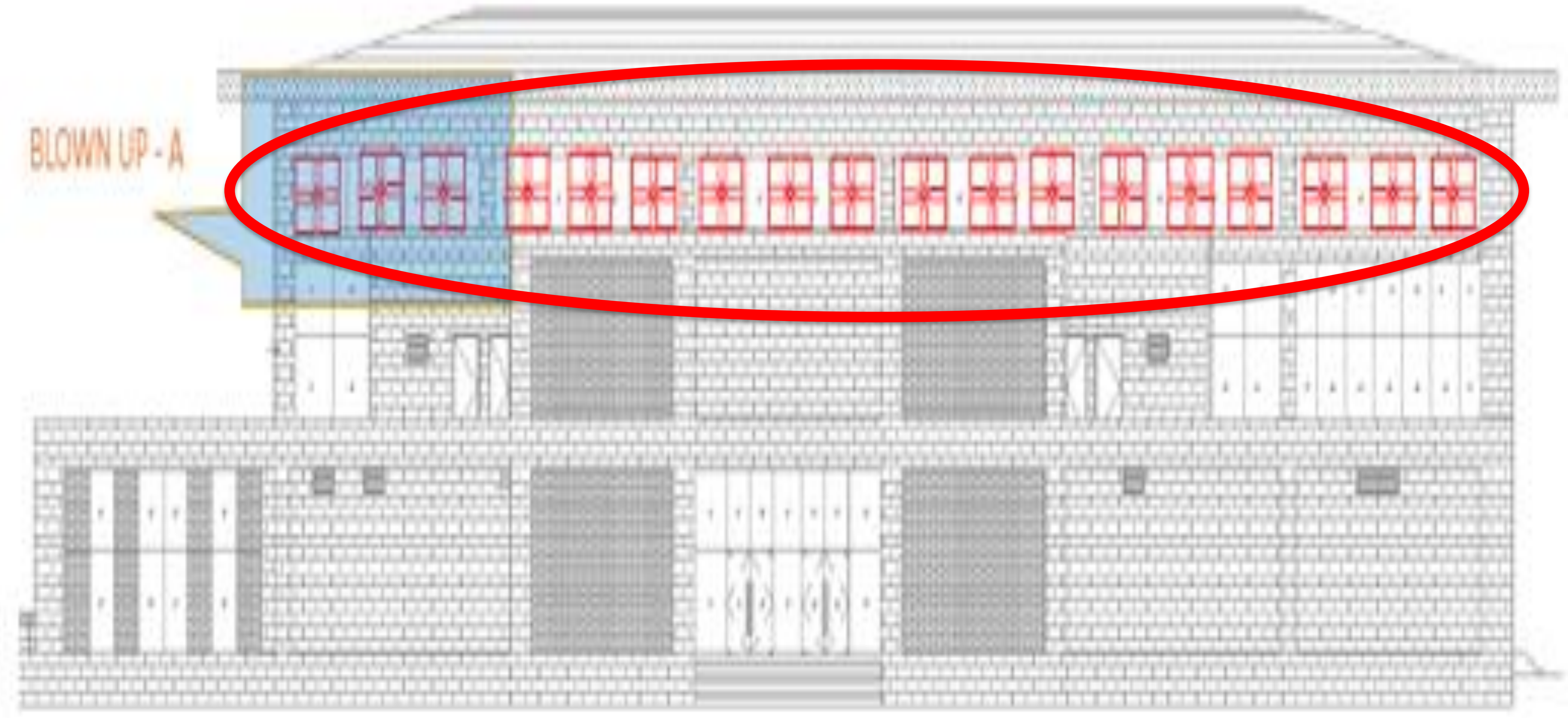
Scenario 2- Velocity profile at 1m

Similar airflow pattern as in Scenario 1 with elongated velocity stream

Scenario 2– Velocity profile at 4m

Airflow velocity values exceed 0.15 m/s as in Scenario 1





CAD model– Scenario 3

Ventilation air is drawn through louvers (circled)

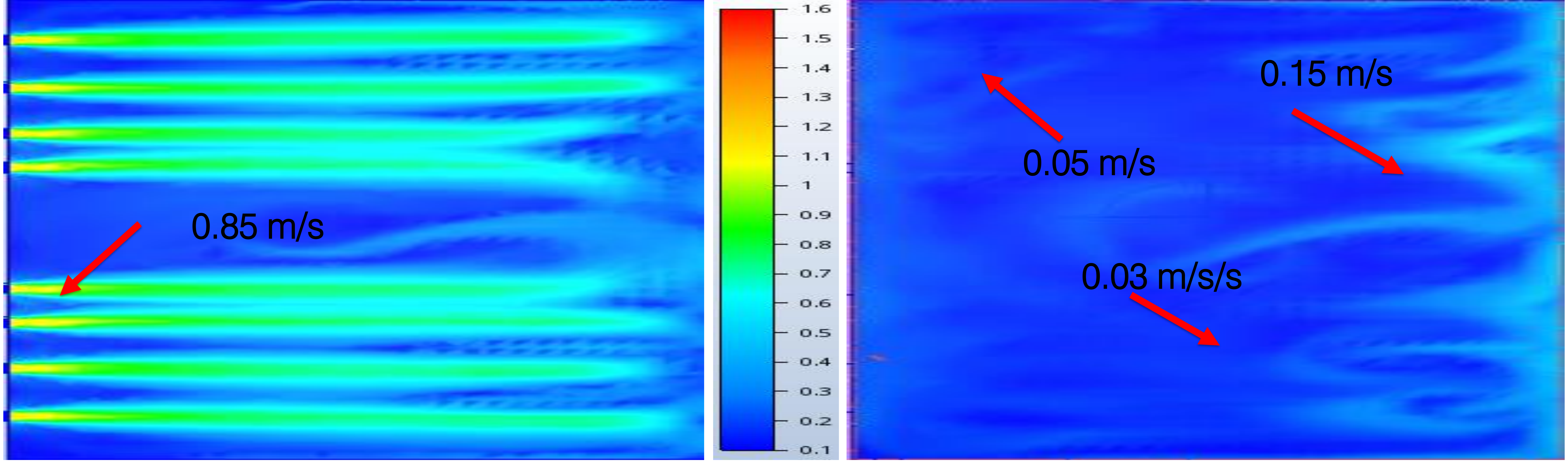
CAD– Scenario 3

Ventilation air movement is driven by propeller fans near roof

Scenario 3- Propeller fans optimized to 4 ACH

Outlet surface of extended air volume is assigned “Static gage pressure =0” as BC

Propeller fans are modelled as CAD part and assigned internal fan material with CFM outwards



Scenario 3- Velocity profile at 1m

Airflow velocity profile at level of 1m from the floor is streamlined without any recirculation

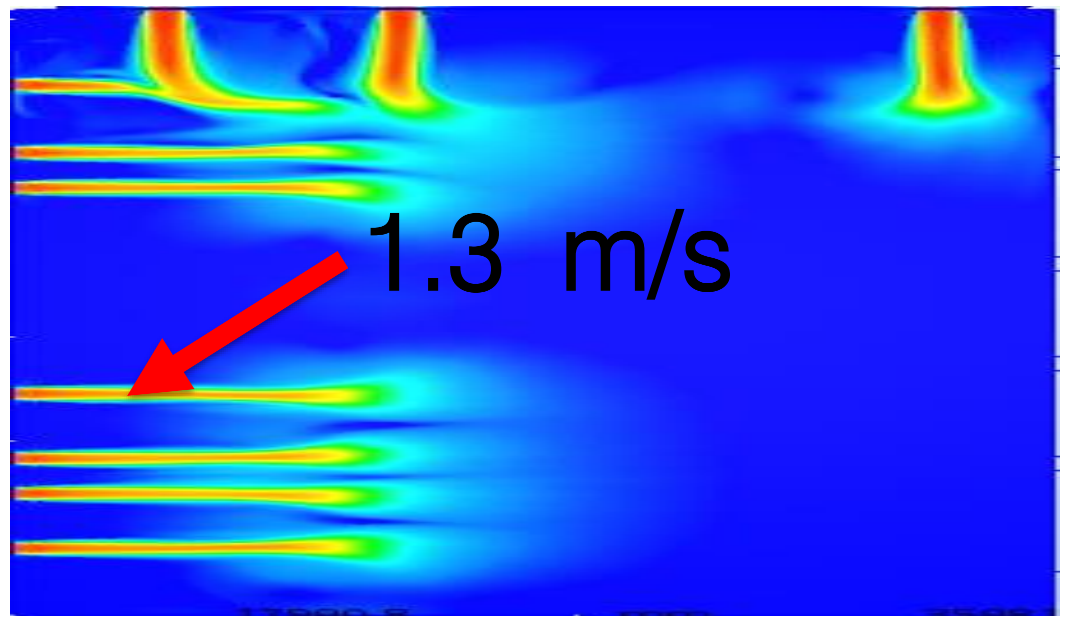
Scenario 3– Velocity profile at 4m

Airflow velocity at height of 4m from the floor does not exceed 0.15 m/s in the playing zone

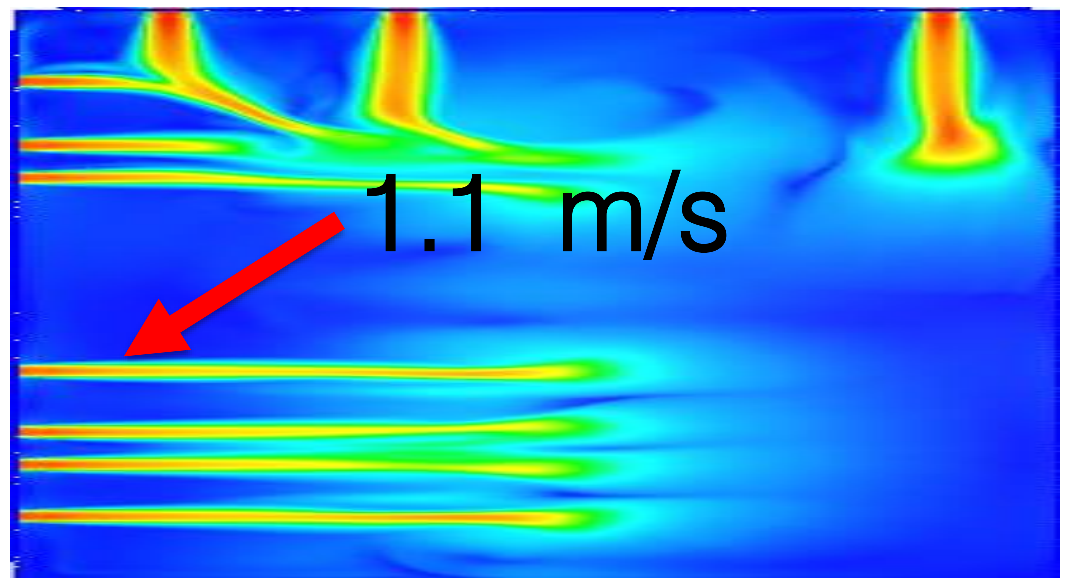
Key Outcome

CFD results at 1m and 4 m level from floor are compared for all 3 scenarios w.r.t acceptance criterion

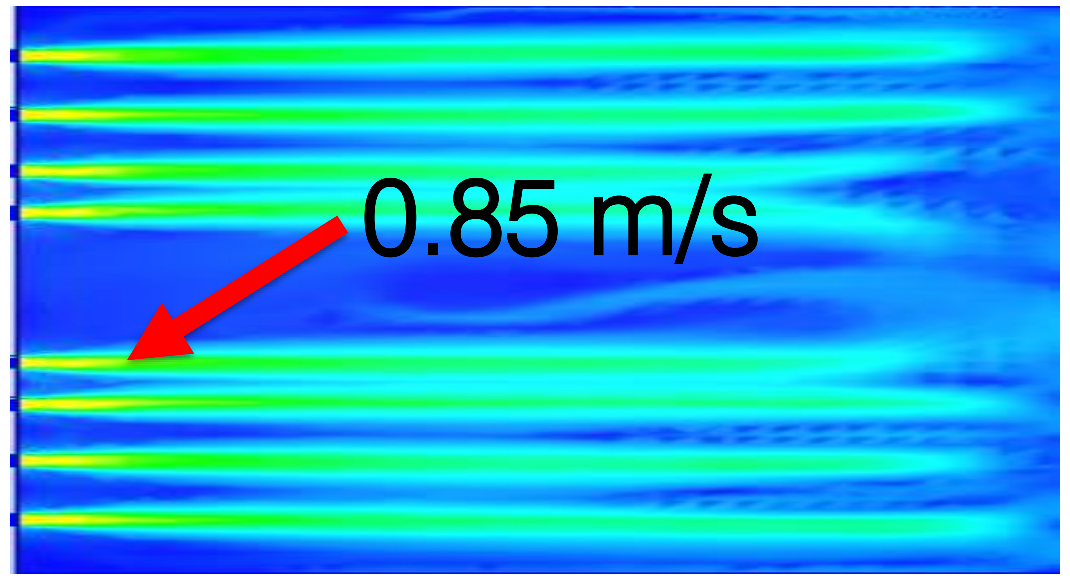
- Scenario 1: airflow velocities are higher than recommended values
- Scenario 2: airflow velocities are higher than recommended values
- Scenario 3: airflow velocities do not exceed 0.15m/s in the playing zone and hence acceptable.
  - Scenario 3 results in energy savings of 26% and hence energy efficient
    - Also cost-effective alternative to more energy consuming air- conditioning system, so meeting sustainable design goals



Scenario 1



Scenario 2

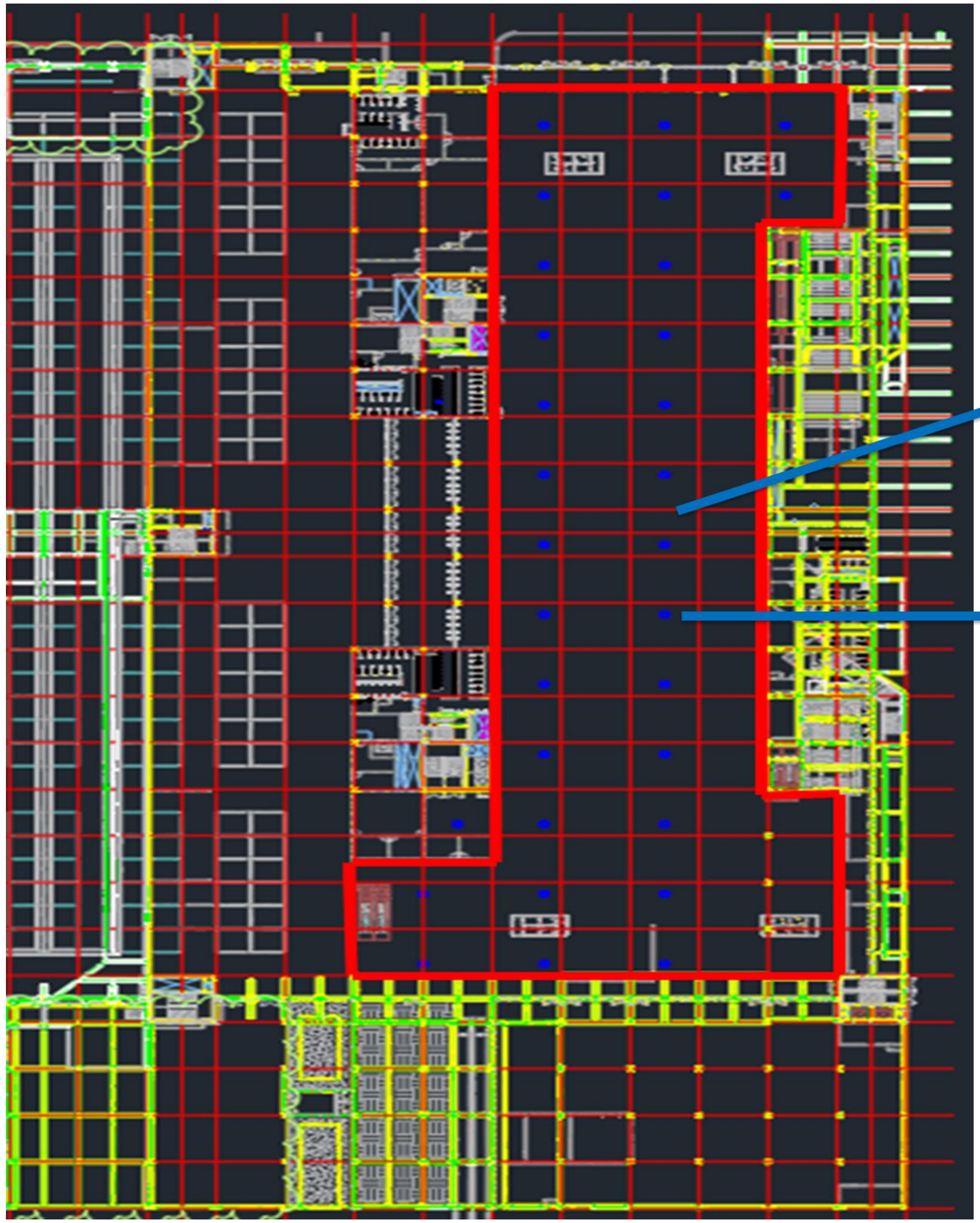


Scenario 3

(ii) Convention Center Foyer

Displacement ventilation (DV) system is cost effective for maintaining thermal comfort and reduced cooling costs compared to traditional mixing air-conditioning systems, thus providing benefits of sustainable design

- **Autodesk CFD Simulation can be used effectively for performance based design evaluation of DV system.**
  - Grand Foyer area of convention center building is marked in “red” in the plan view shown in the right.
    - The Grand Foyer is a large open space for circulation that gives access to the exhibition halls, and houses cafes, restaurants, meeting rooms, etc. and to the Convention Centre.



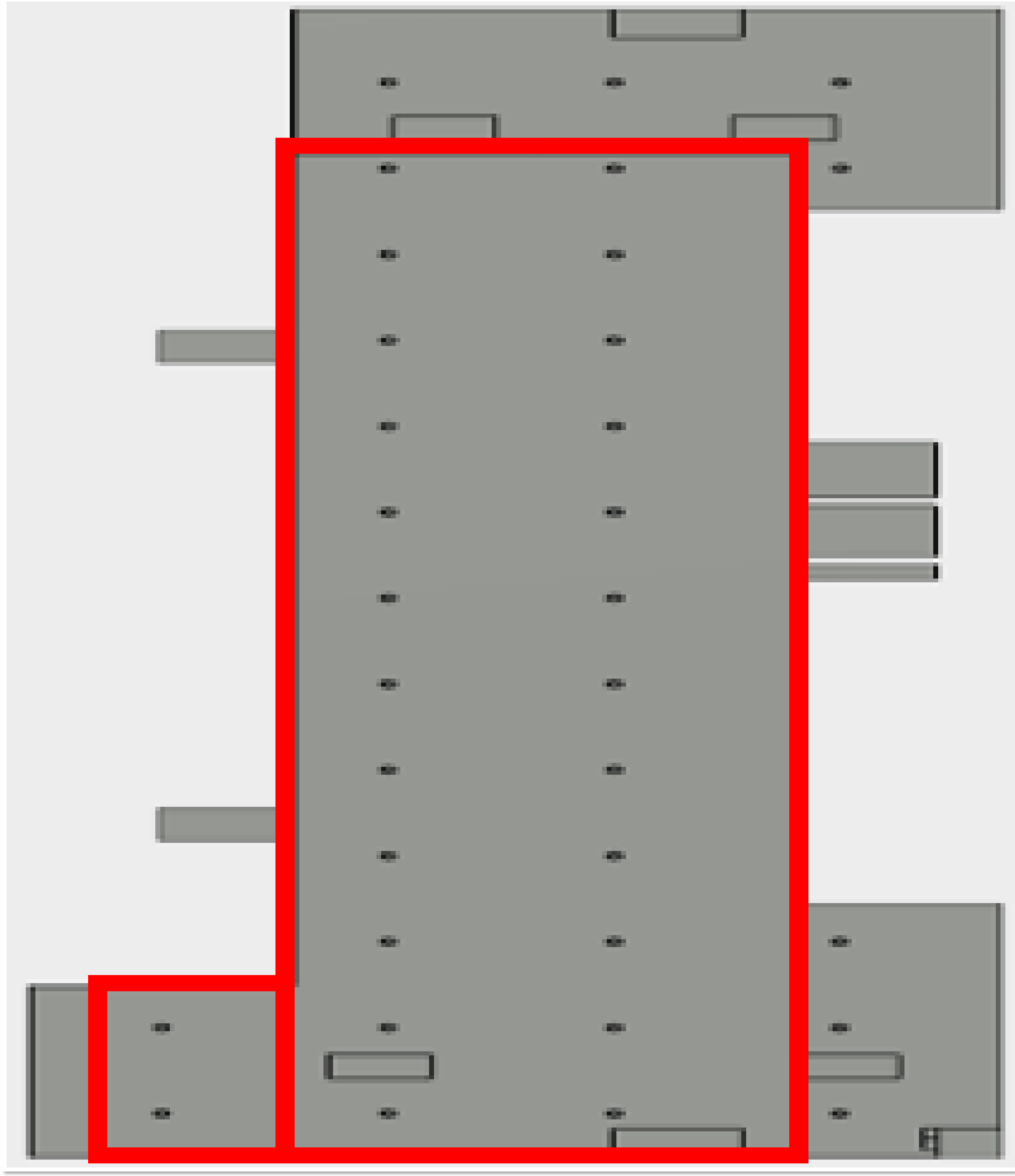
Foyer
Area = 82075 sq.ft

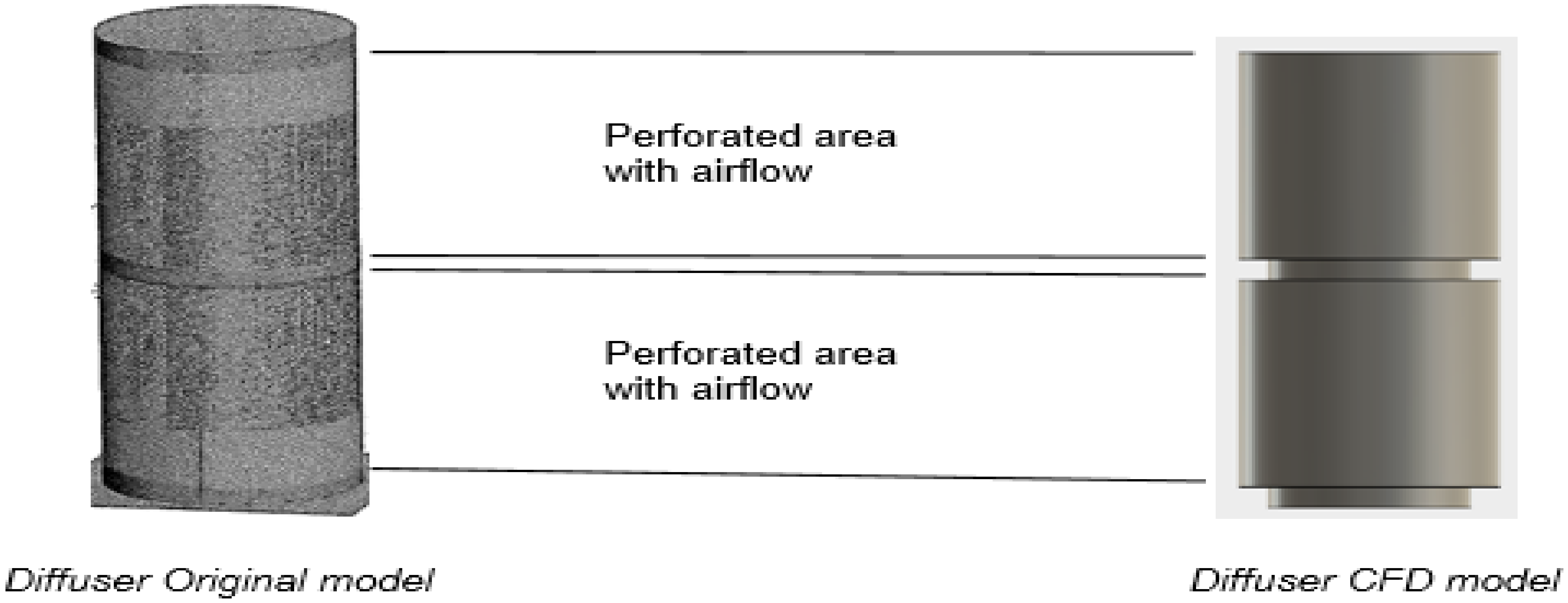
DV diffusers

Heat load details

Plan view of CAD model of Foyer with marked area for heat load distribution

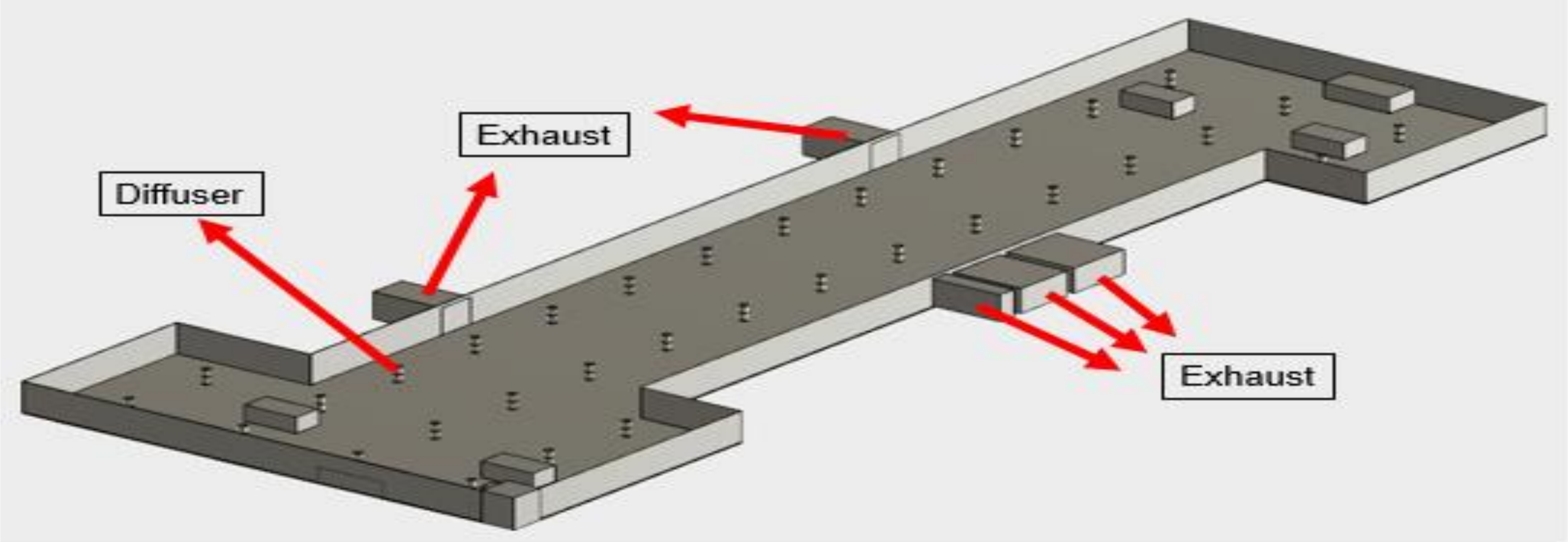
- Lighting load = 1.9 W/ Sq. ft.
- Equipment load = 82 kW
- Occupancy = 16.0 Sq. ft. /person
- Occupancy, No. of persons = 5160





Diffuser component characterization

Displacement tower diffuser produces a low velocity redial airflow pattern from perforated cylindrical discharge, low-level lake of high quality supply air that floods the floor with airflow. Energy saving also accrue the extended free cooling range achieved by the elevated supply air temperature as well as from the potential to reduce outdoor airflow rate due to the enhanced indoor air quality resulting from the improved ventilation effectiveness of the low-level displacement supply.



CFD model with supply and return details

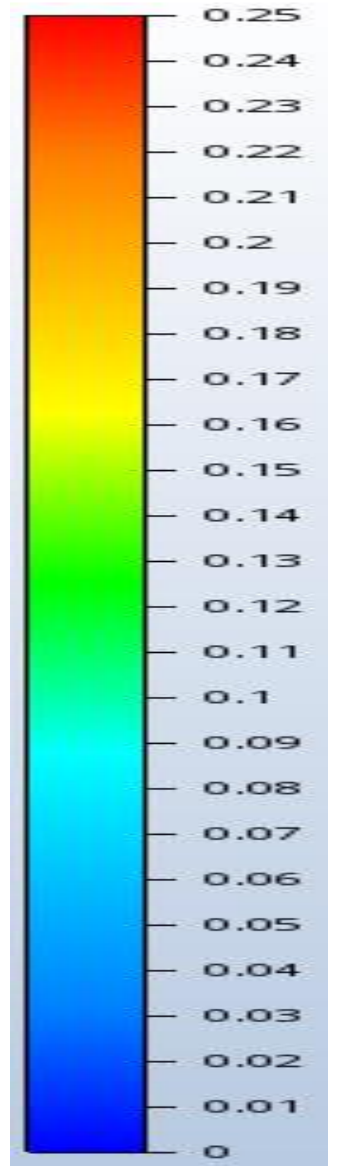
Diffuser inlet flow rate = 4000 - 4500 CFM; Diffuser inlet temperature = 14^0 C; Diffuser free area ratio = 0.7

Exhaust Pressure outlet = 0 Pa (gauge)

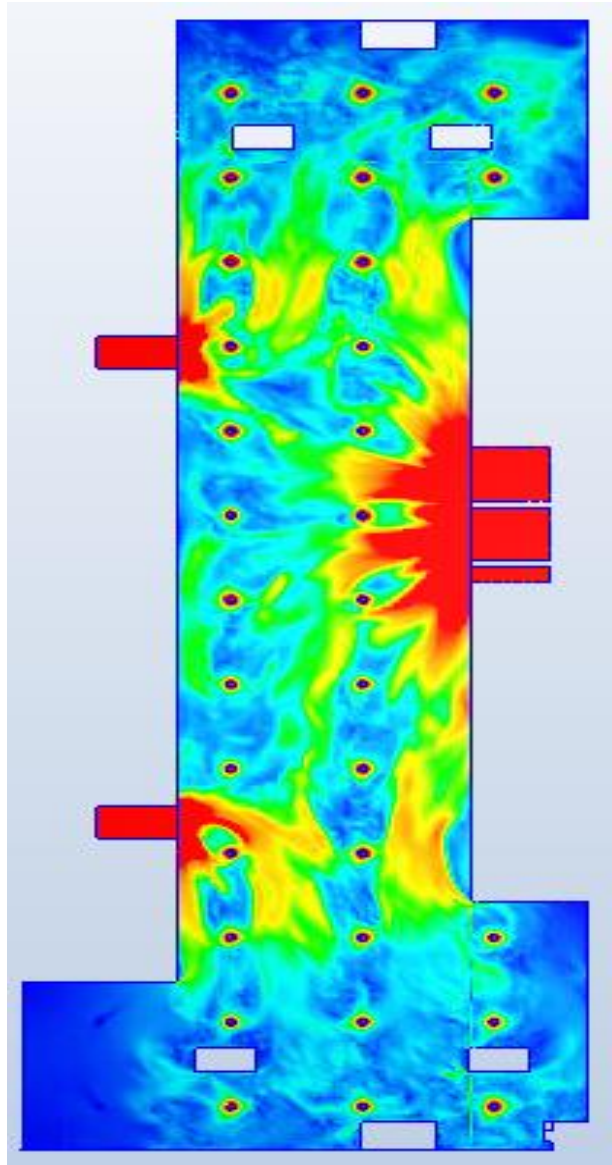
Exhaust louver free area ratio = 0.6

Inside design condition = 24^0 C

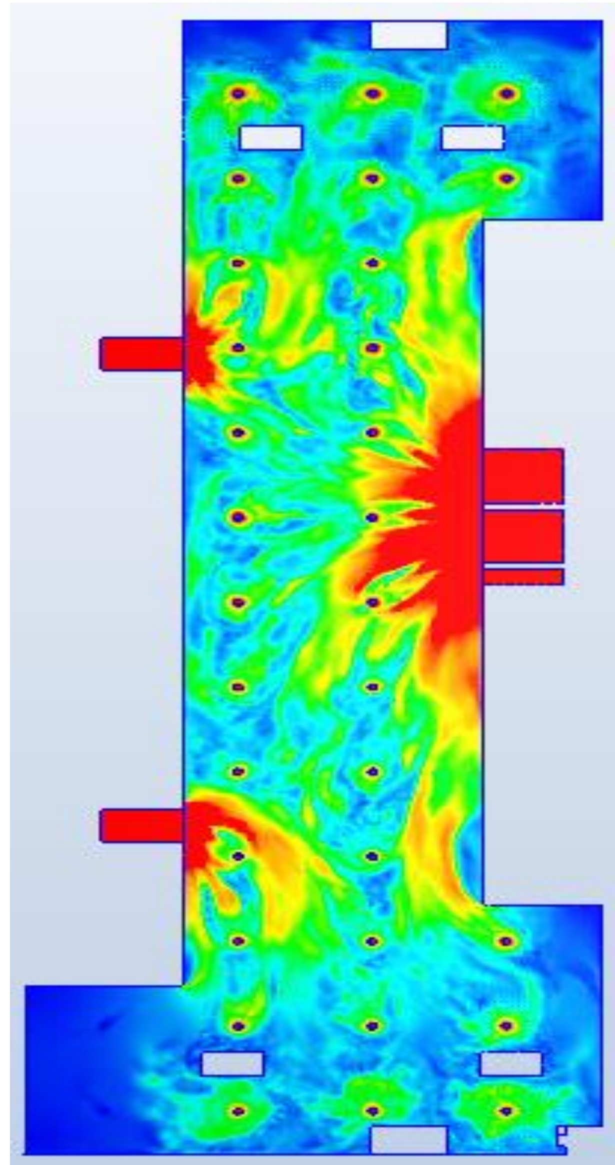
Total heat load = Human load + Equipment load = (5160 * 71) W + (82000) W



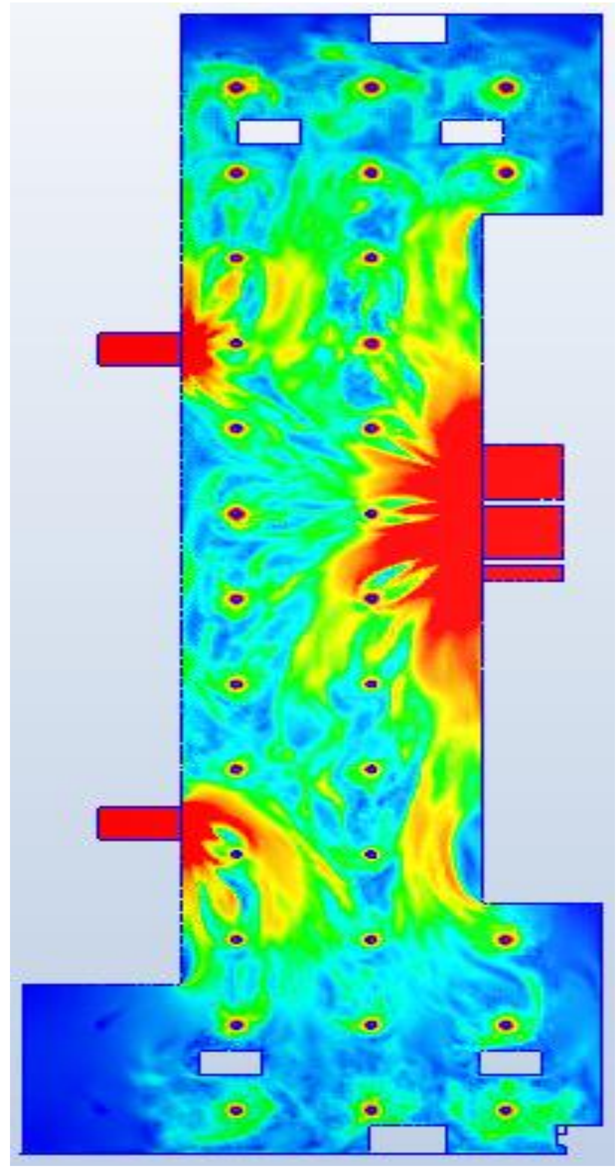
Velocity, m/s



Velocity contour at 0.7 m from the floor



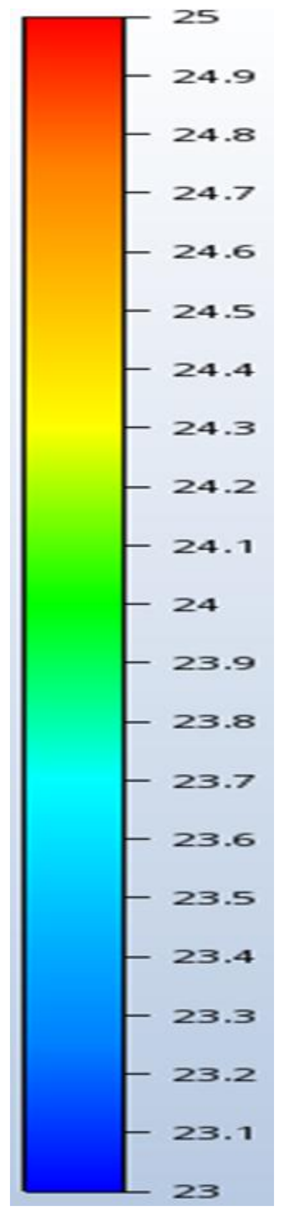
Velocity contour at 1.7 m from the floor



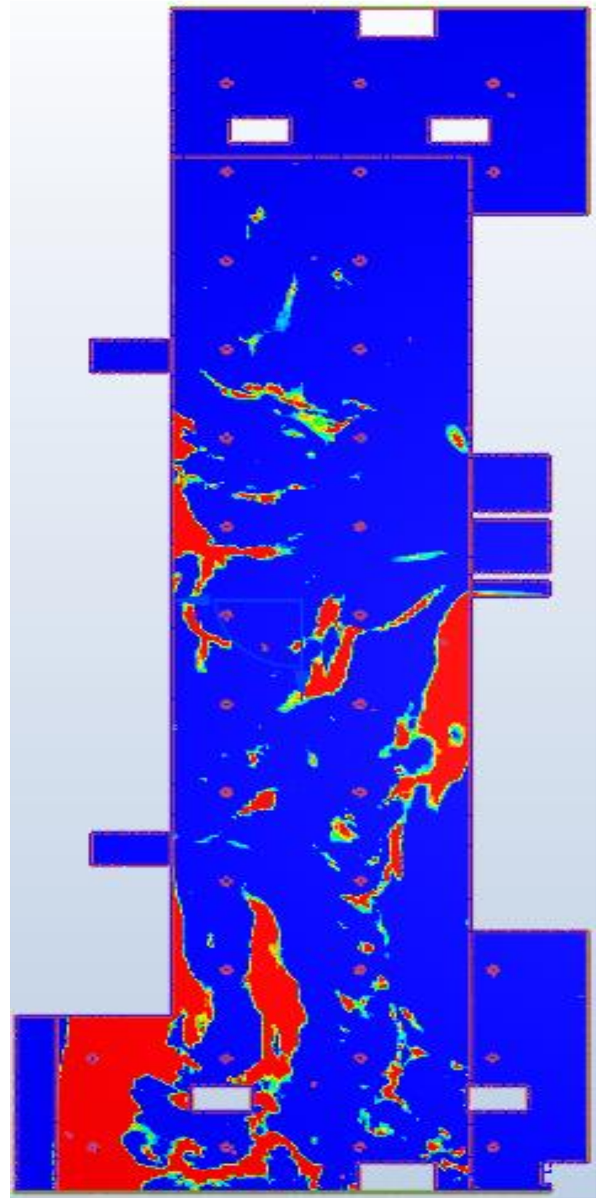
Velocity contour at 2.0 m from the floor

Air flow distribution

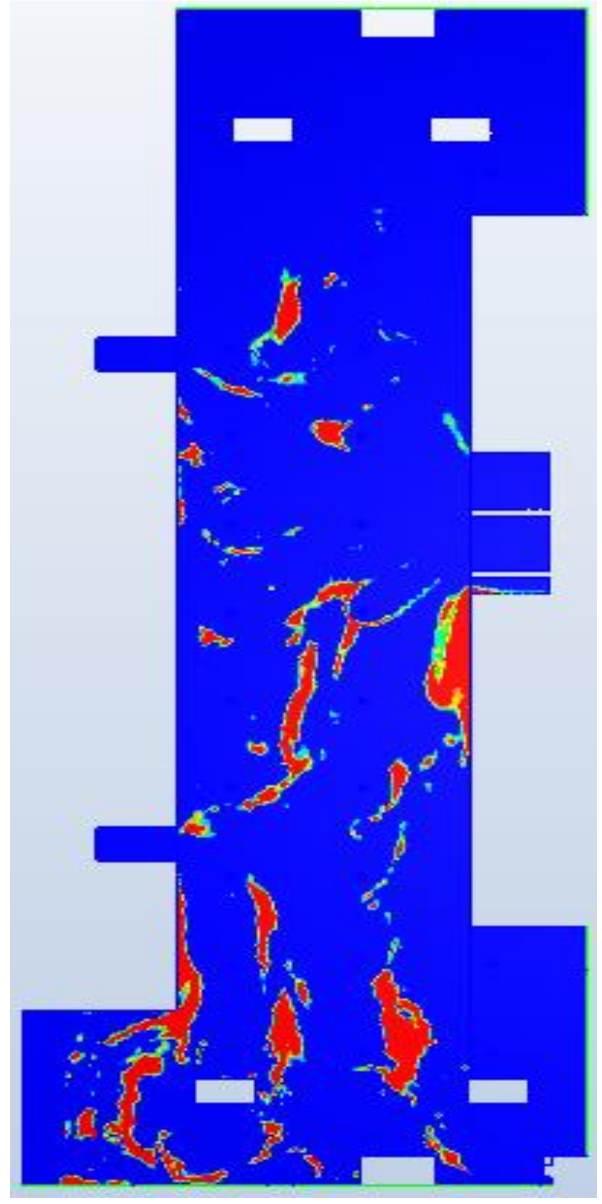
The velocity contour shows a clear distribution of airflow all over the foyer area except the corners, which are not critical areas for people movement.



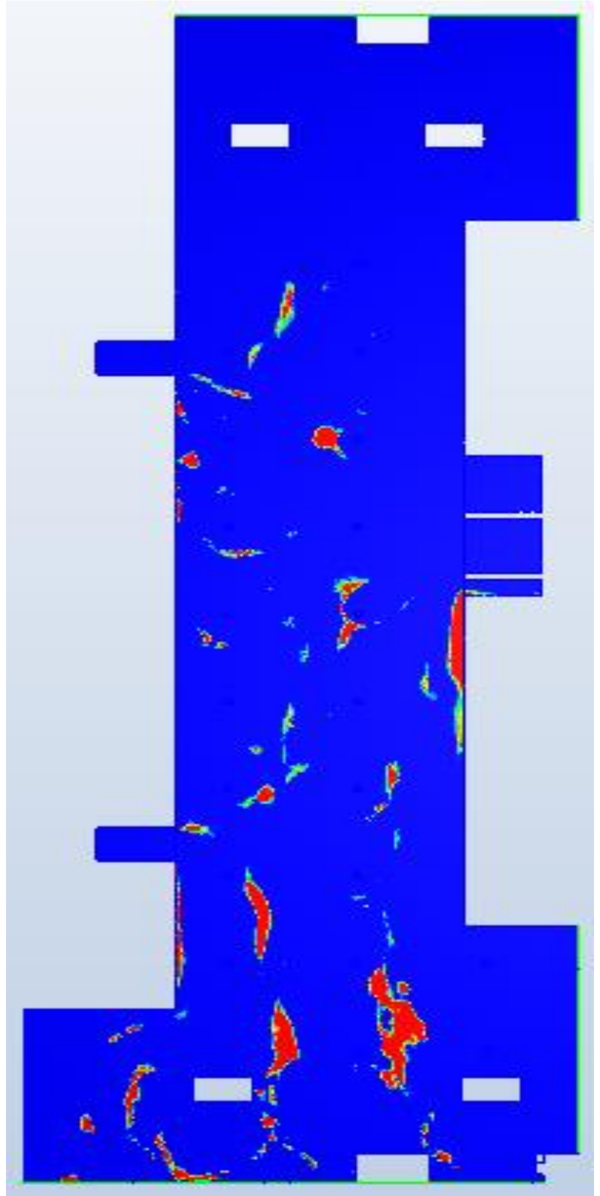
Temperature ^{0}C



Temperature contour at 0.7 m from the floor



Temperature contour at 1.7 m from the floor



Temperature contour at 2.0 m from the floor

Temperature distribution

Temperature distribution shows occupied areas are within acceptable limit. There are certain regions where there is slightly high temperature due to poor airflow; however, due to nature of people movement within foyer, it will eventually be neutralized as people keep walking all around the foyer area redistributing heat.

Key outcome

Autodesk CFD Simulation and CFD results are used for:

- performance based design of air-conditioning system
- addressing challenges imposed by architectural/ structural requirements
- evaluating thermal comfort of occupied space (e.g. Foyer) using Displacement Ventilation (DV)
- means of meeting sustainability goals by using energy efficient cooling system such as DV

Wind Flow / Wind Loading

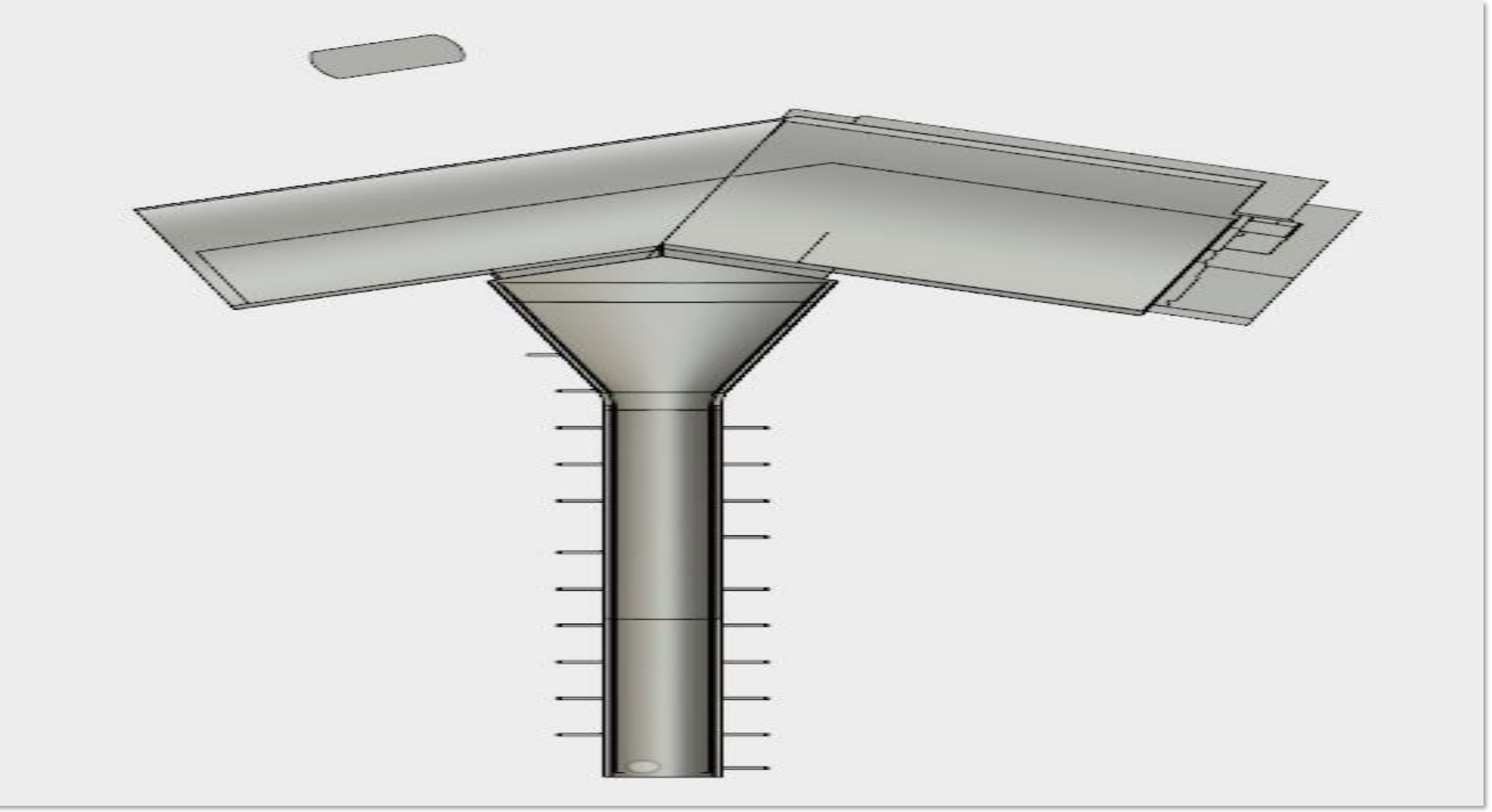
Wind Flow / Wind Loading

Modeling and simulation of wind flow and rainf low on airport buildings

i. Wind loading on Façade and roof of large airport
   - Wind pressure
     - Structural loading

ii. Rainwater ingress into forecourt of a large airport
   - Wind driven flow
     - Rain ingress

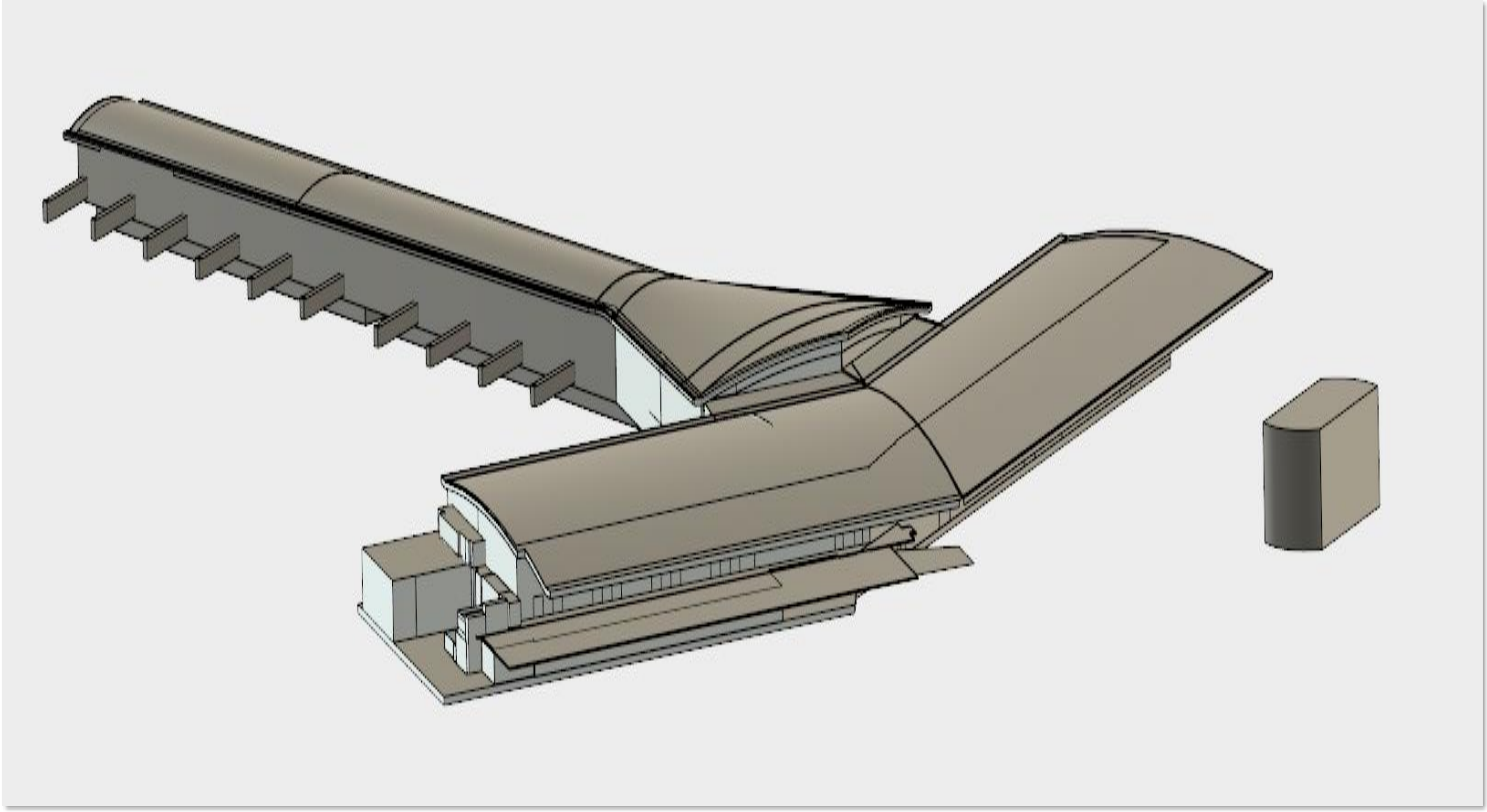
(i) Wind loading



Simulation ready CAD model

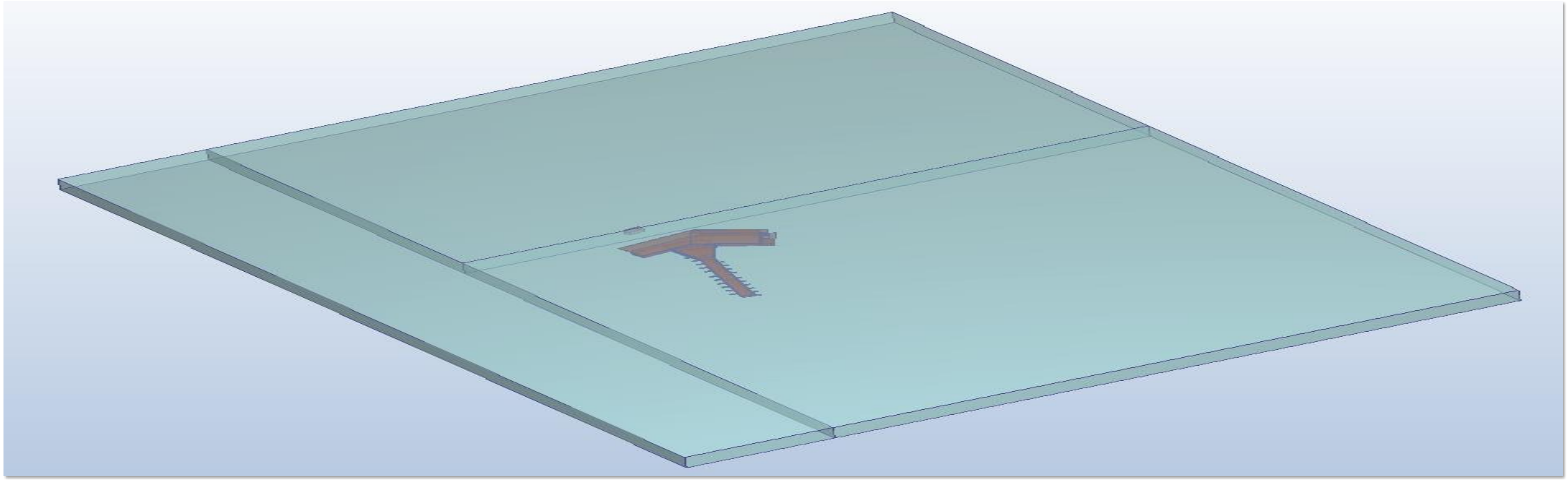
CAD model prepared using Autodesk SimStudio Tools 2016 R3

Geometry simplified by removal of details not relevant to analysis.



Simulation ready CAD model

Simplified geometry without losing design intent results in efficient CFD simulation in terms of time and hardware requirements

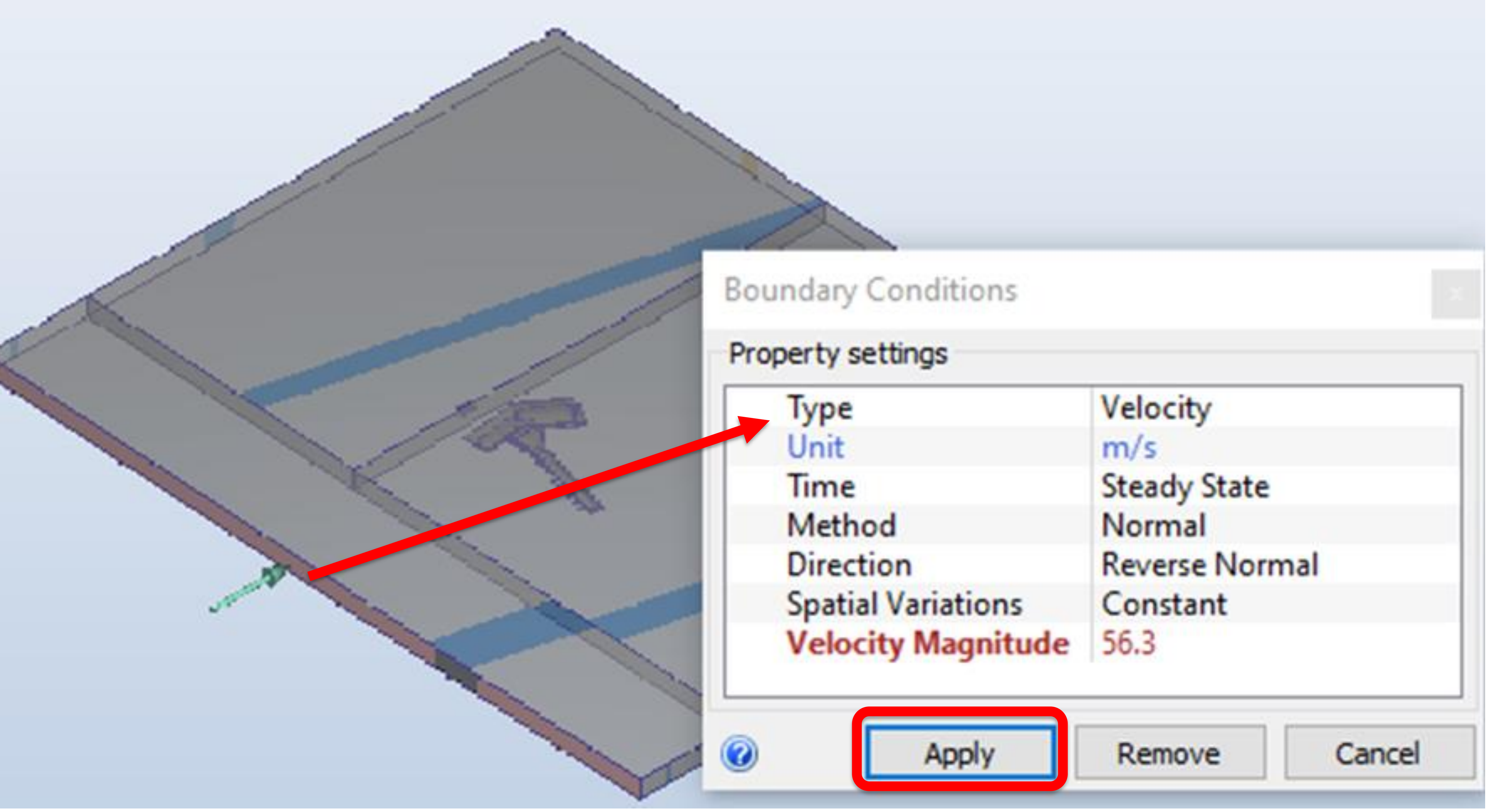


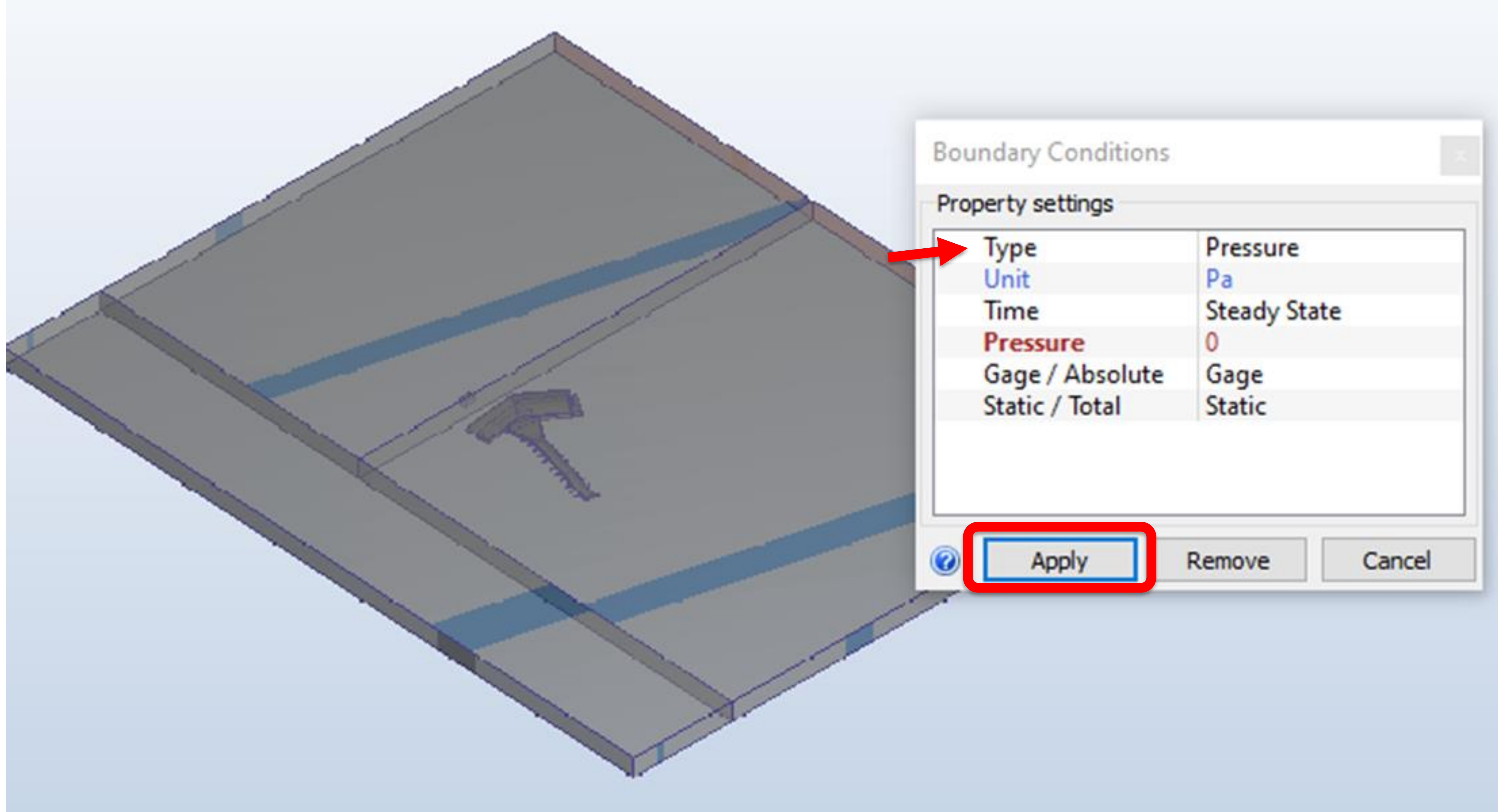
CFD model

External volume surrounding the building is added to the model using the CAD tool.

External volume geometry is built using Autodesk recommended dimensions

External volume is assigned “Air” material with constant density



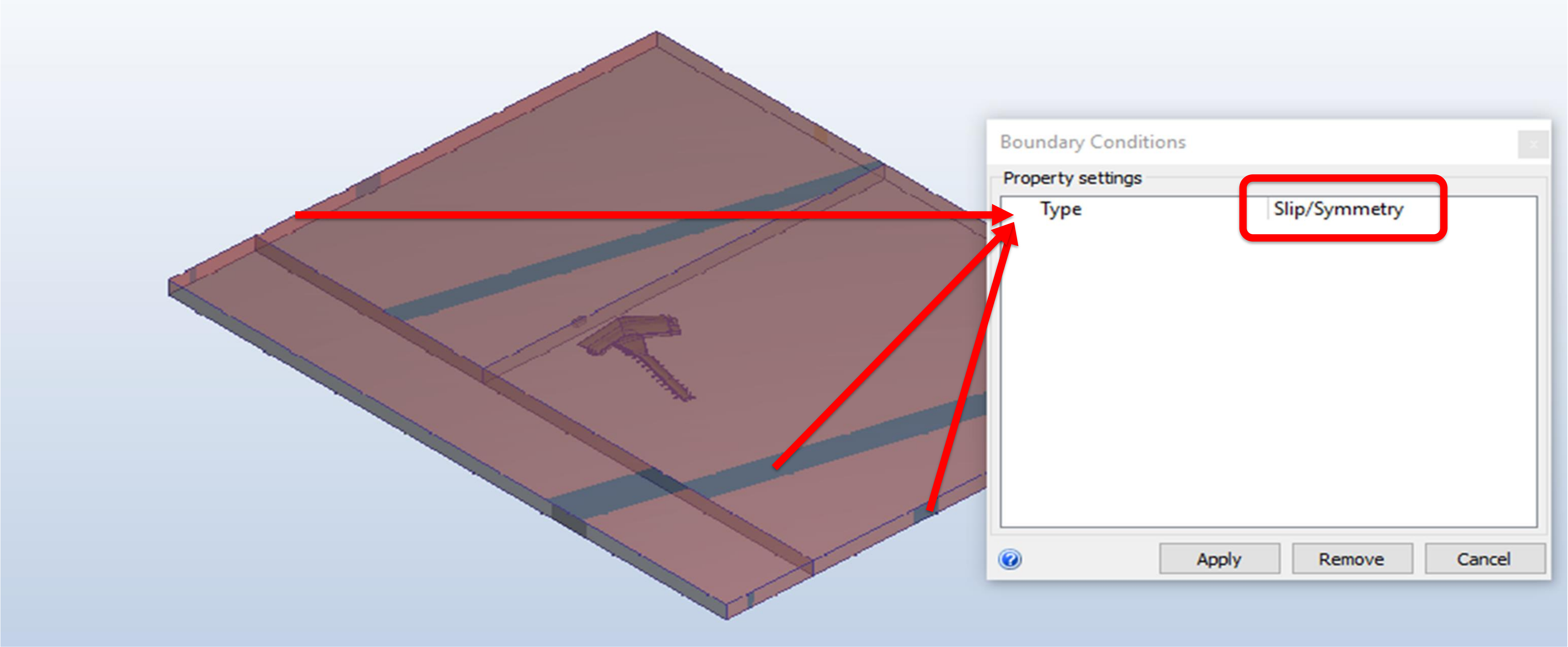


CFD model– Inlet BC

Wind speed is assigned as velocity BC at the inlet of air volume.

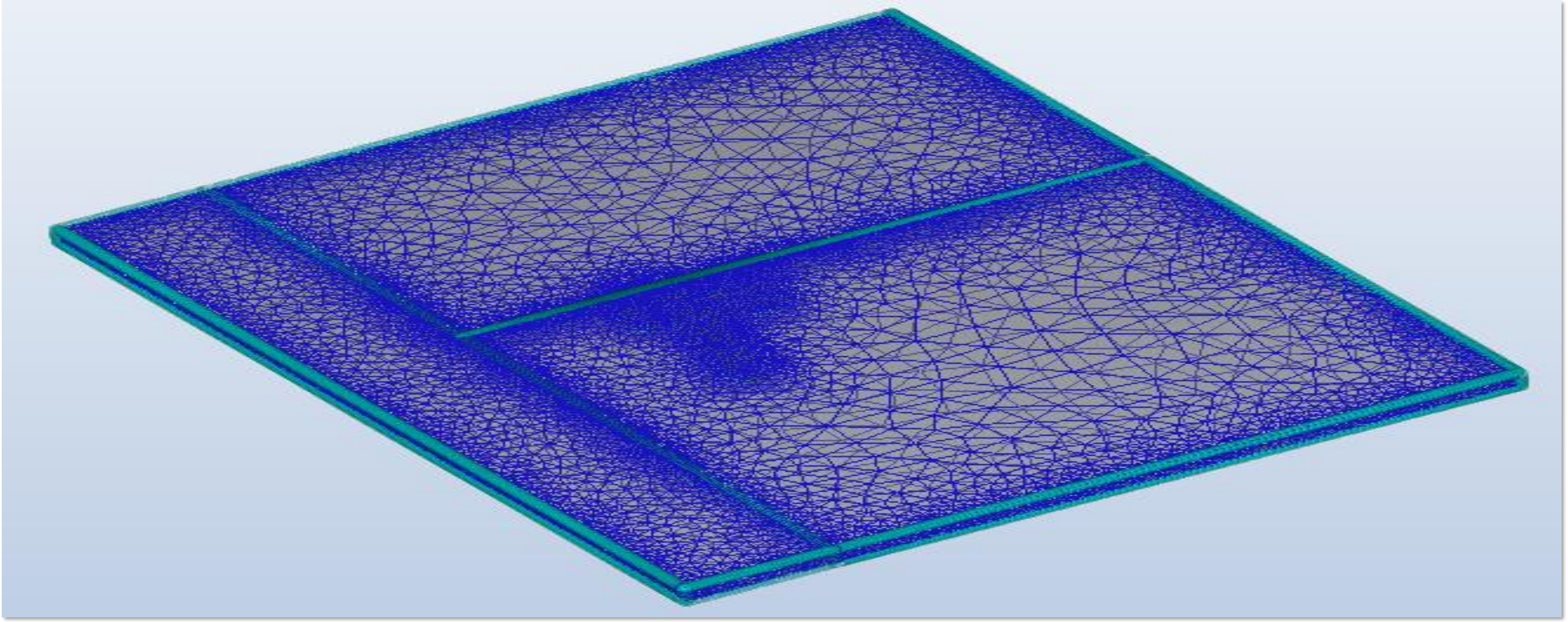
CFD model– Outlet BC

Outlet of air volume is assigned static gage pressure = 0



CFD model – Free environment BC

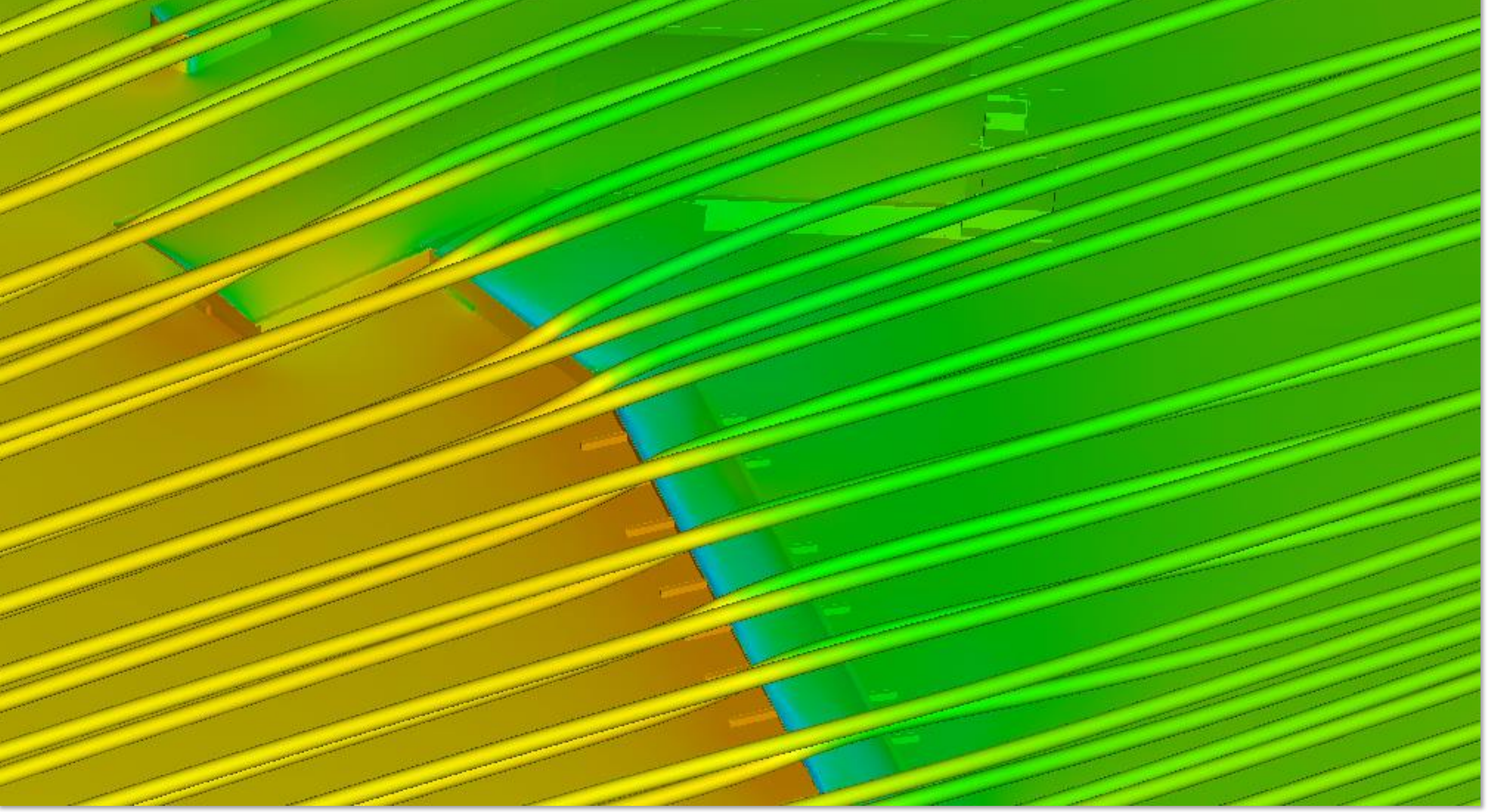
External air volume is assigned slip/symmetry BC as it is a free space

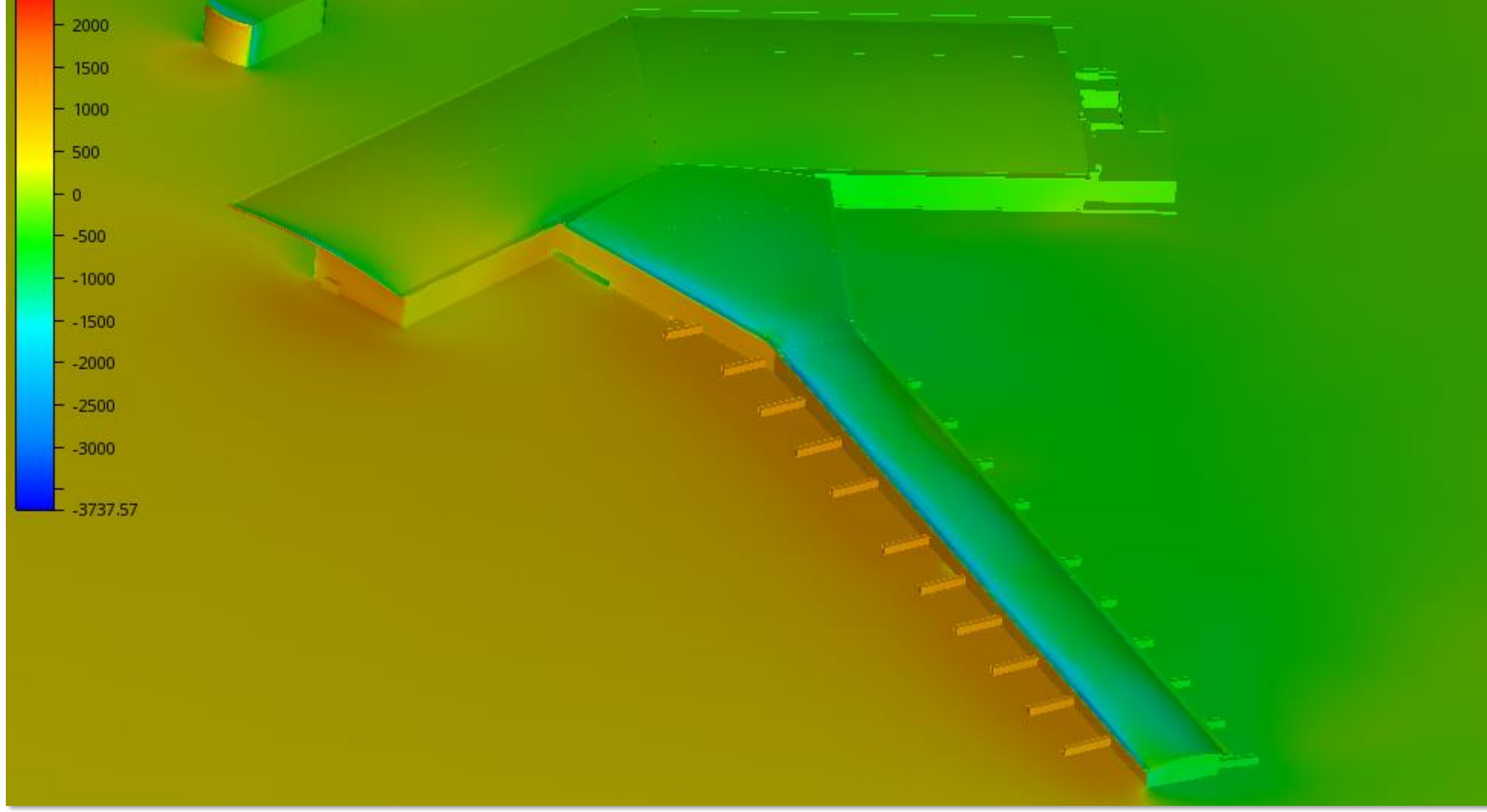


CFD model – Mesh

Automatic mesh sizing is used to define mesh distribution in the CFD solution domain

Mesh refinement is applied on the geometric volume to resolve circulating flows/gradients



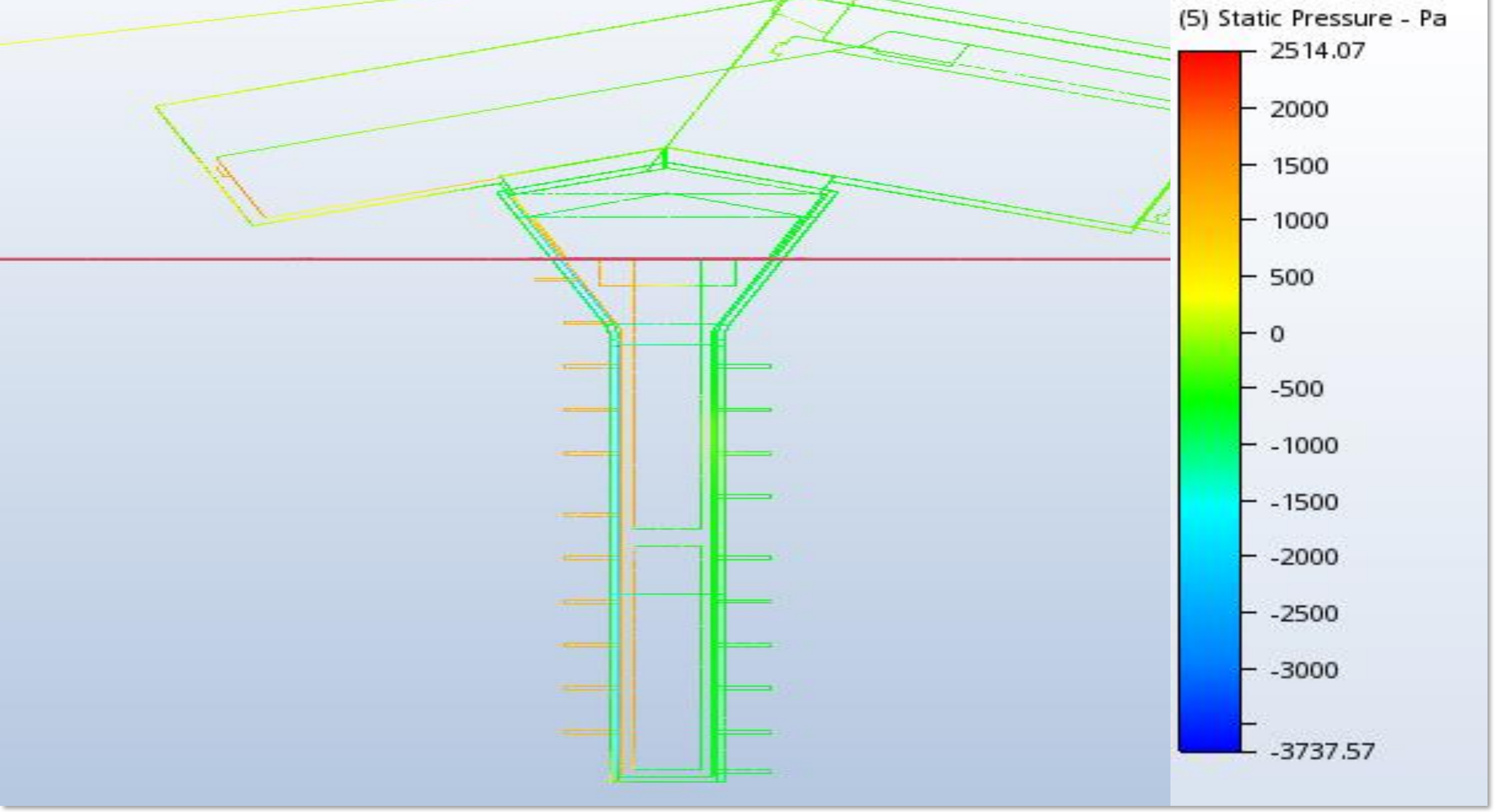


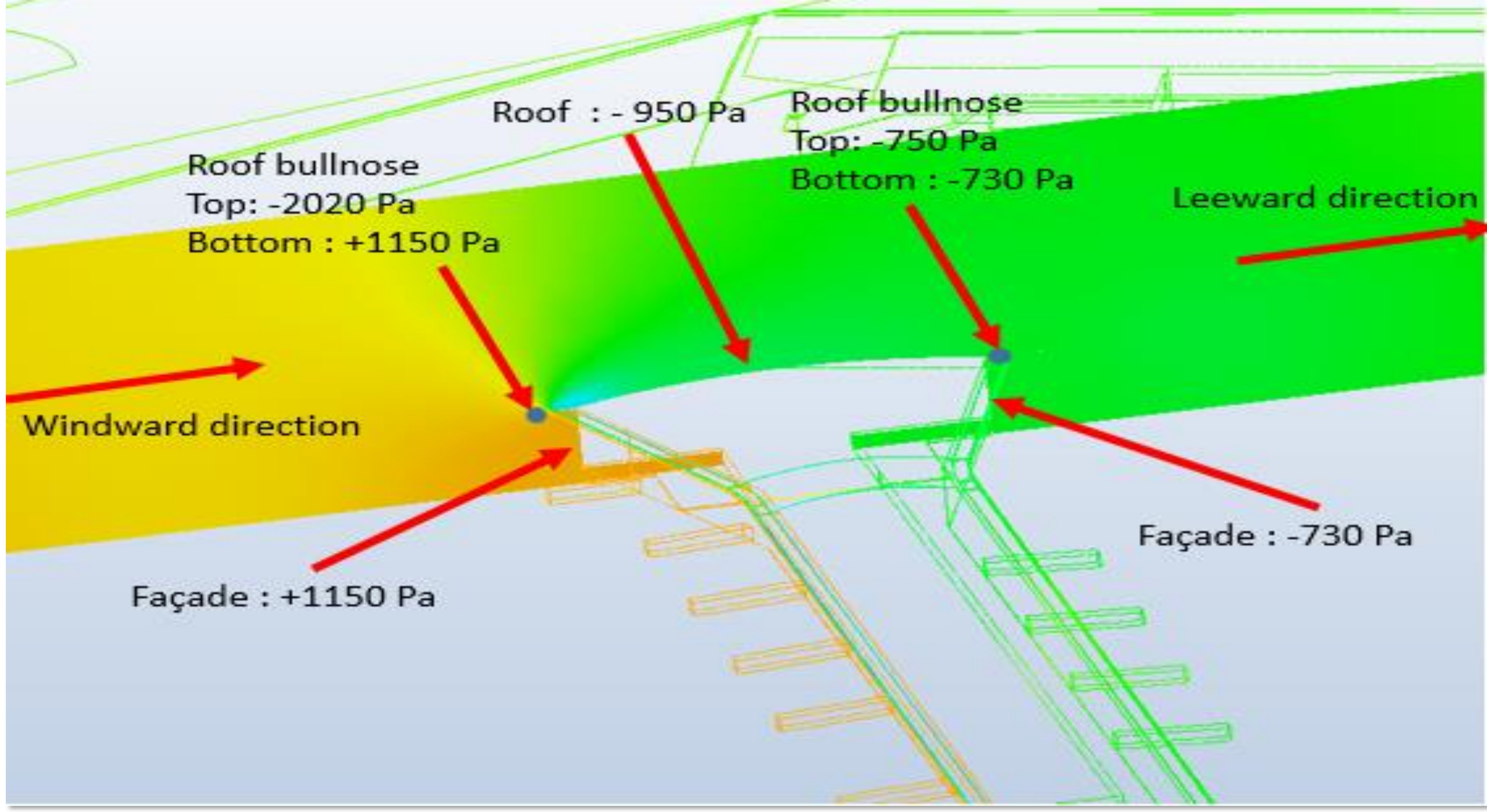
CFD results– Traces

Particle traces to visualise air movement

CFD results– Global pressure values

Global results for static pressure to visualize pressure distribution on the airport façade and roof





CFD results– Planes

Using Results Planes option to extract results

CFD results–Pressure values

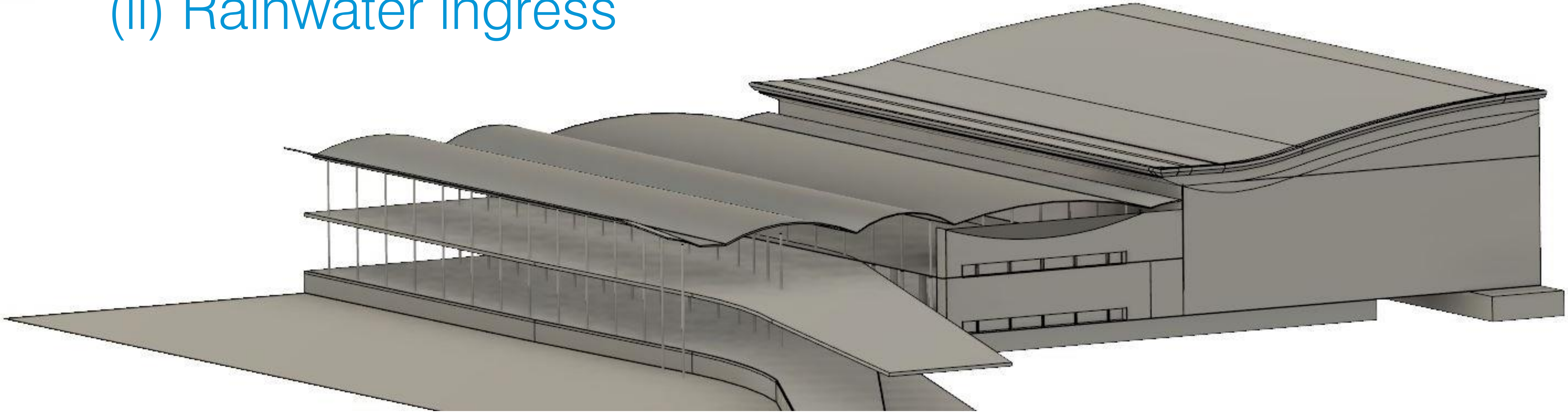
Static pressure values at location of interest to evaluate wind loading effect

Key outcome

Autodesk CFD Simulation and CFD results provide:

- Useful information on wind flow
- Resultant wind pressure
- Means of evaluating structural loading
- Design adequacy check using CFD results -> less rework and optimised design
- Means of meeting sustainability goals

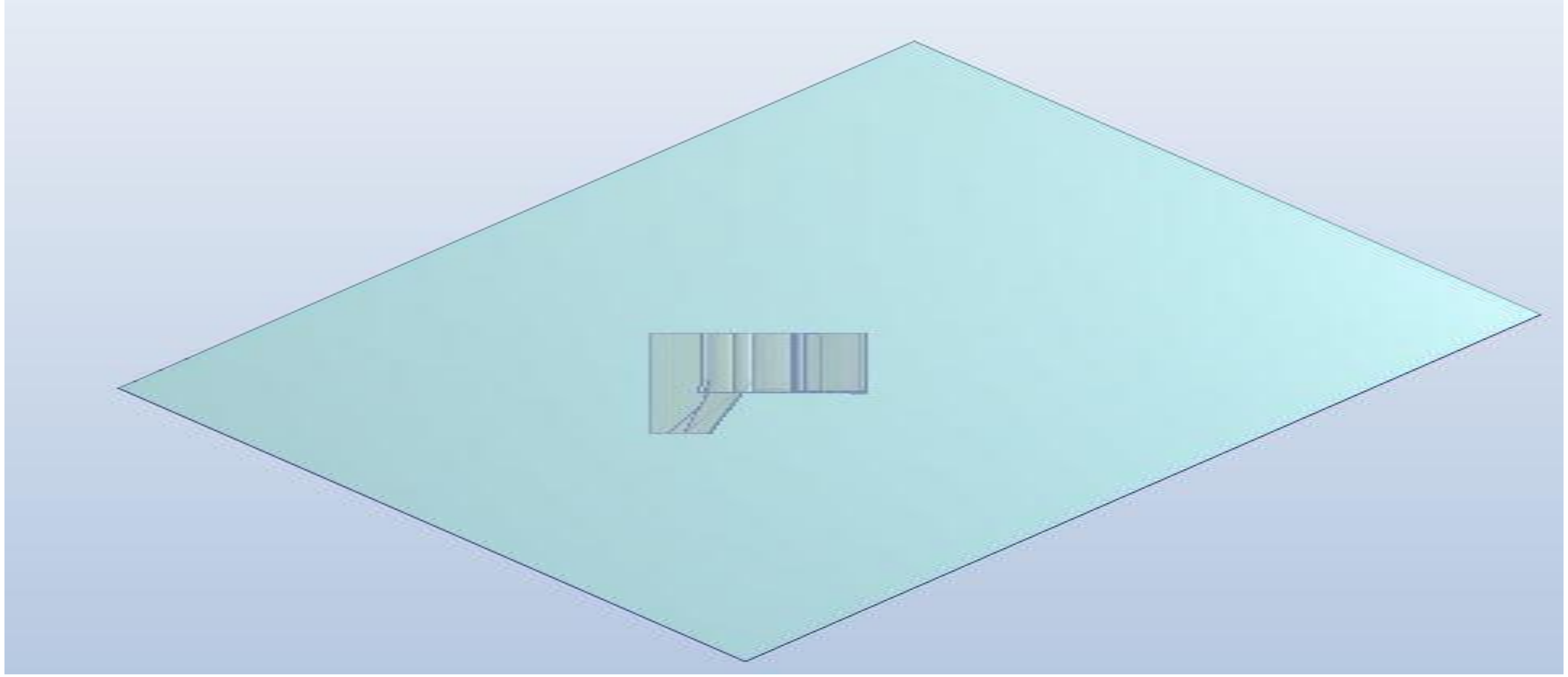
(ii) Rainwater ingress



Simulation ready CAD model

Simulation ready CAD model of airport terminal building with forecourt, prepared using Autodesk Fusion 360

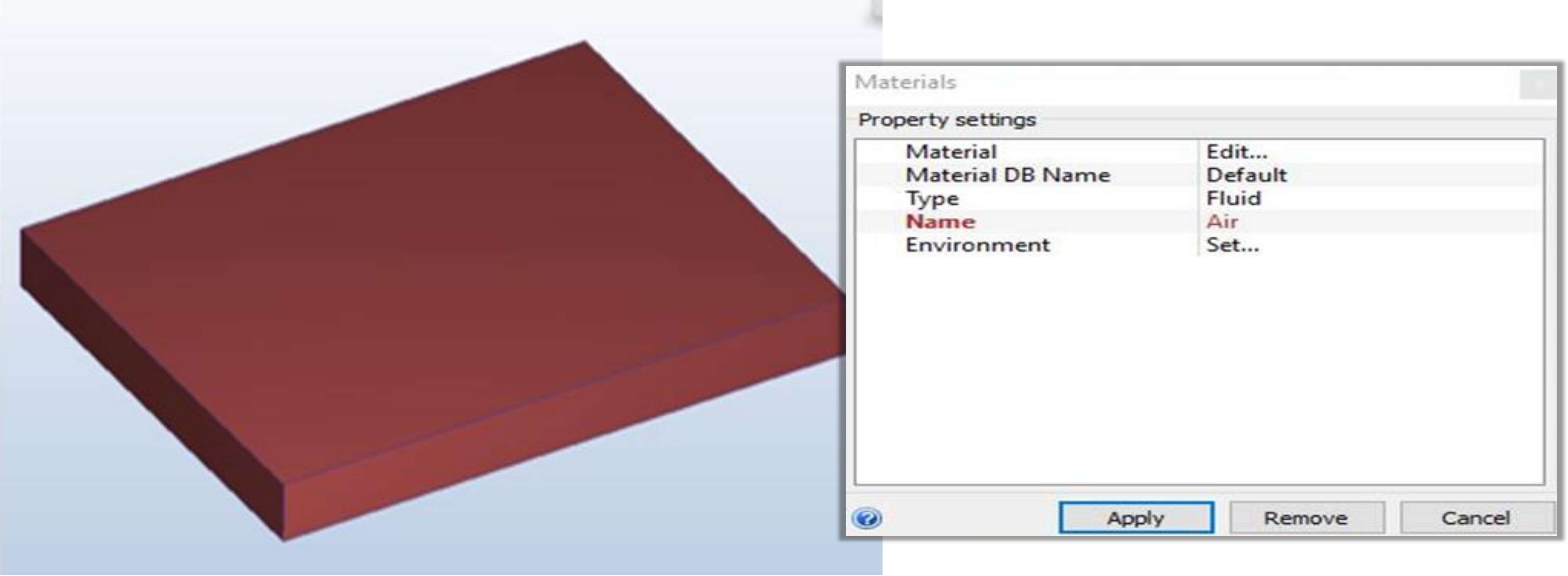
Geometry simplified by removal of details not relevant to analysis, without losing geometry intent of design.



CFD model with external volume

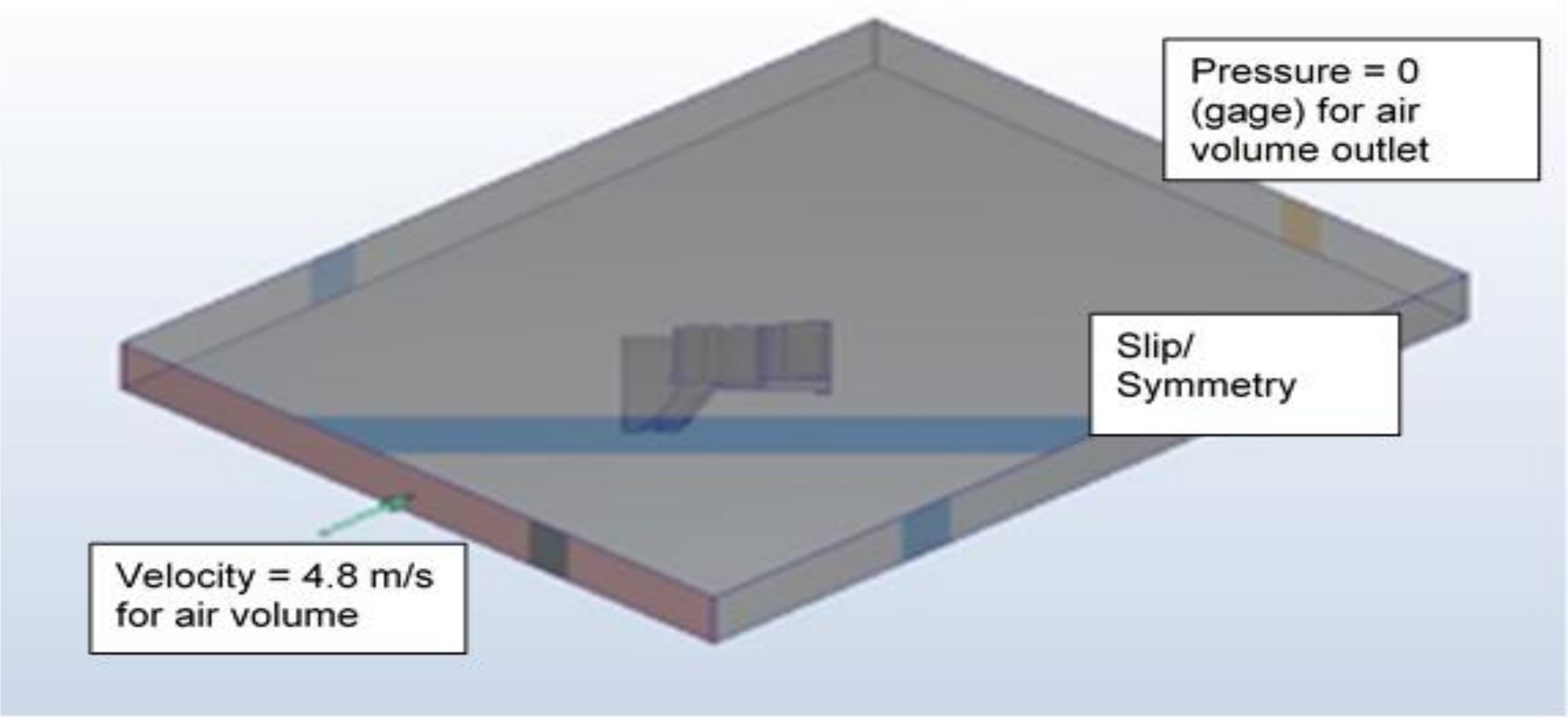
External volume

- added to the model using Autodesk CFD External Volume tool
- built using Autodesk CFD recommended dimensions
- used to define wind flow into and around the building
- assigned “Air” material with constant density



“Air” material assignment in CFD

“Materials” tab from the ribbon of “Setup Tasks” is used to assign “Air” material as “Fluid” to the external air volume

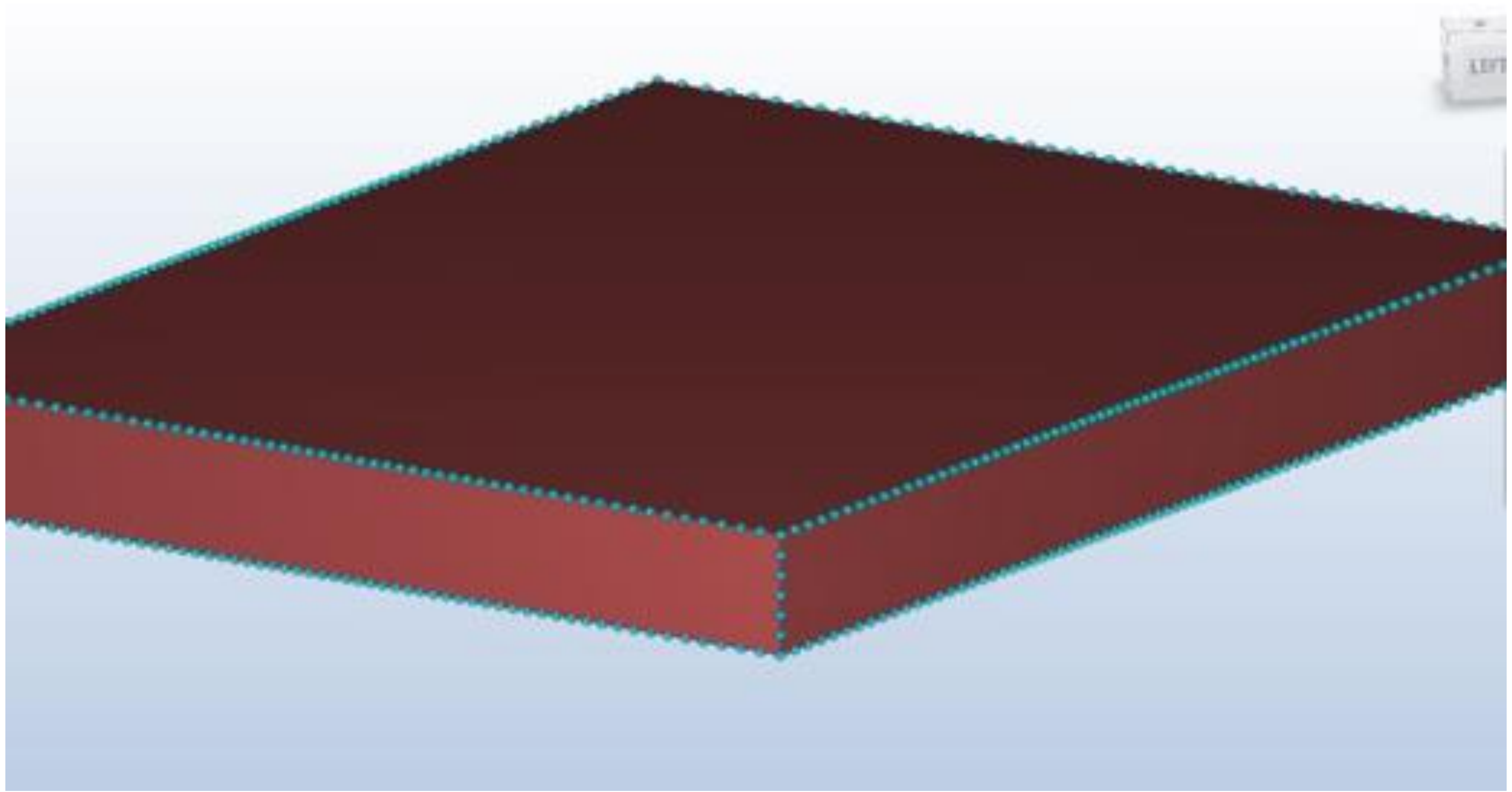


CFD model – Boundary Conditions (BCs)

External air volume inlet face -> Velocity BC

External air volume outlet face -> Pressure BC

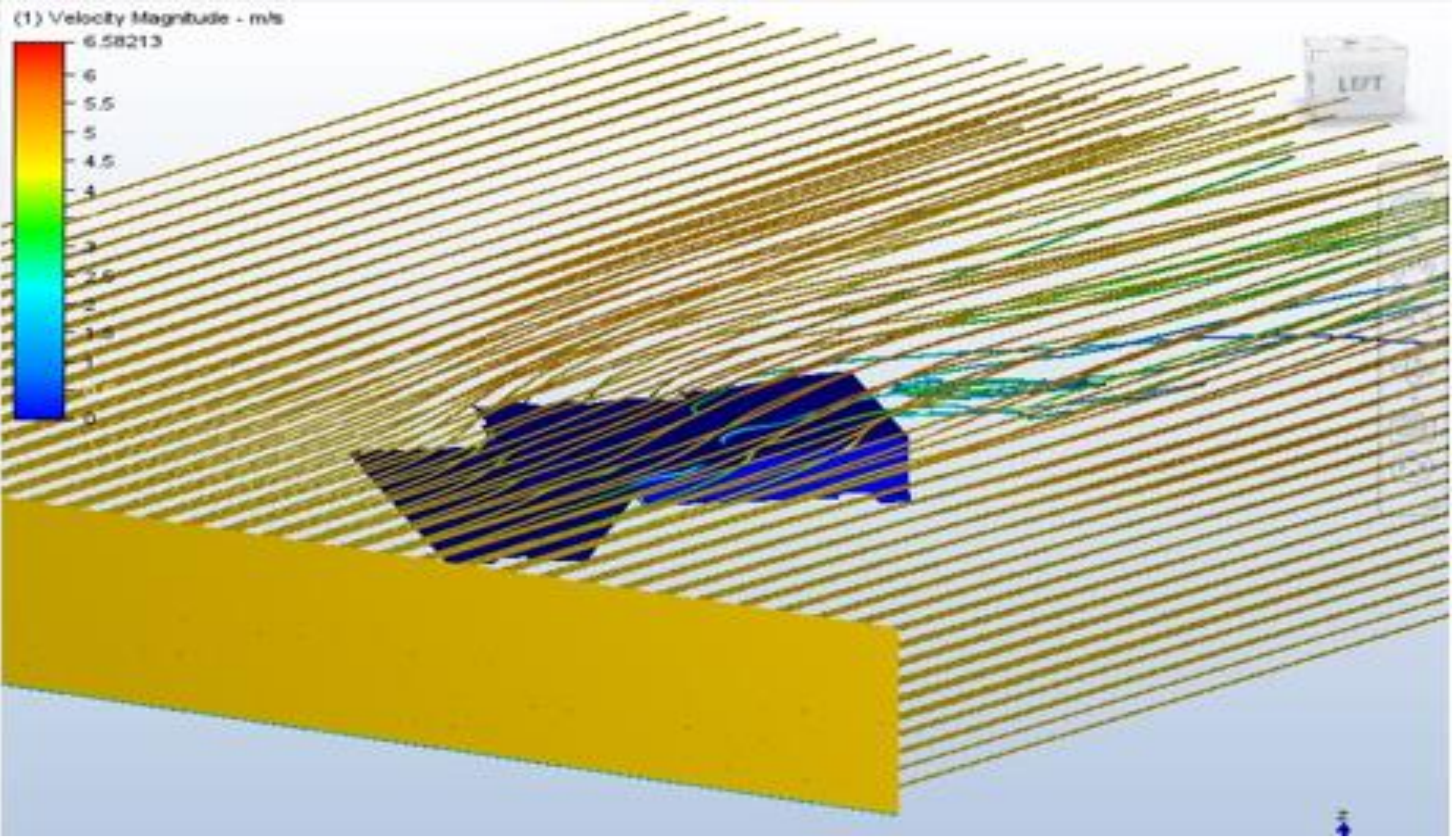
Top and sides of external air volume -> slip/symmetry BC

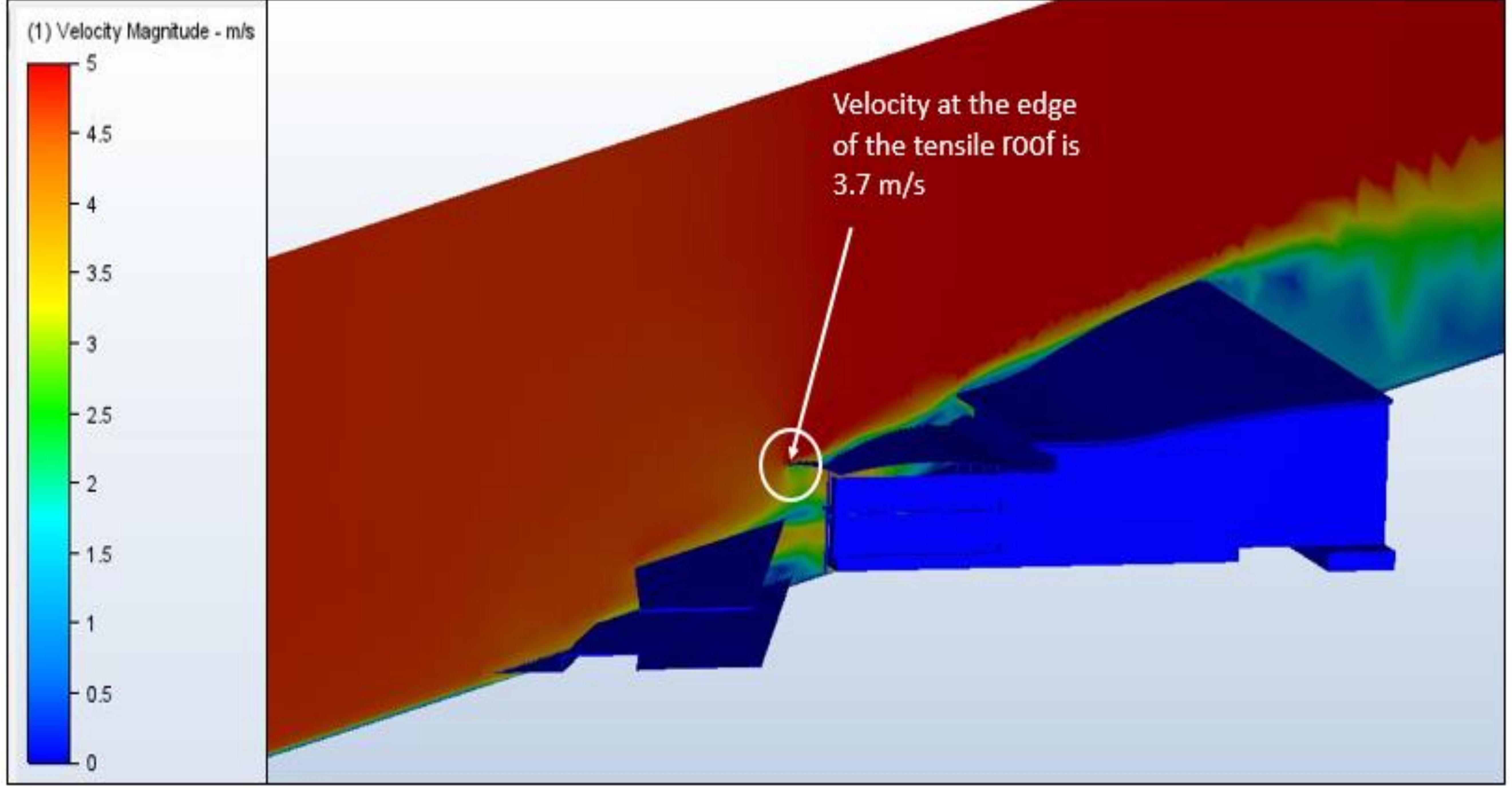


CFD model – Mesh

Automatic mesh sizing is used to define mesh distribution in the CFD solution domain

Mesh refinement is applied on the geometric volume to resolve circulating flows/gradients





CFD results– Traces

Particle traces to visualise air movement

CFD results– Velocity values

Plane results for air flow velocity to visualize flow on the airport roof

Rain deflection angle based on CFD results

Rain deflection angle due to wind speed on the roof edge is calculated using below reference

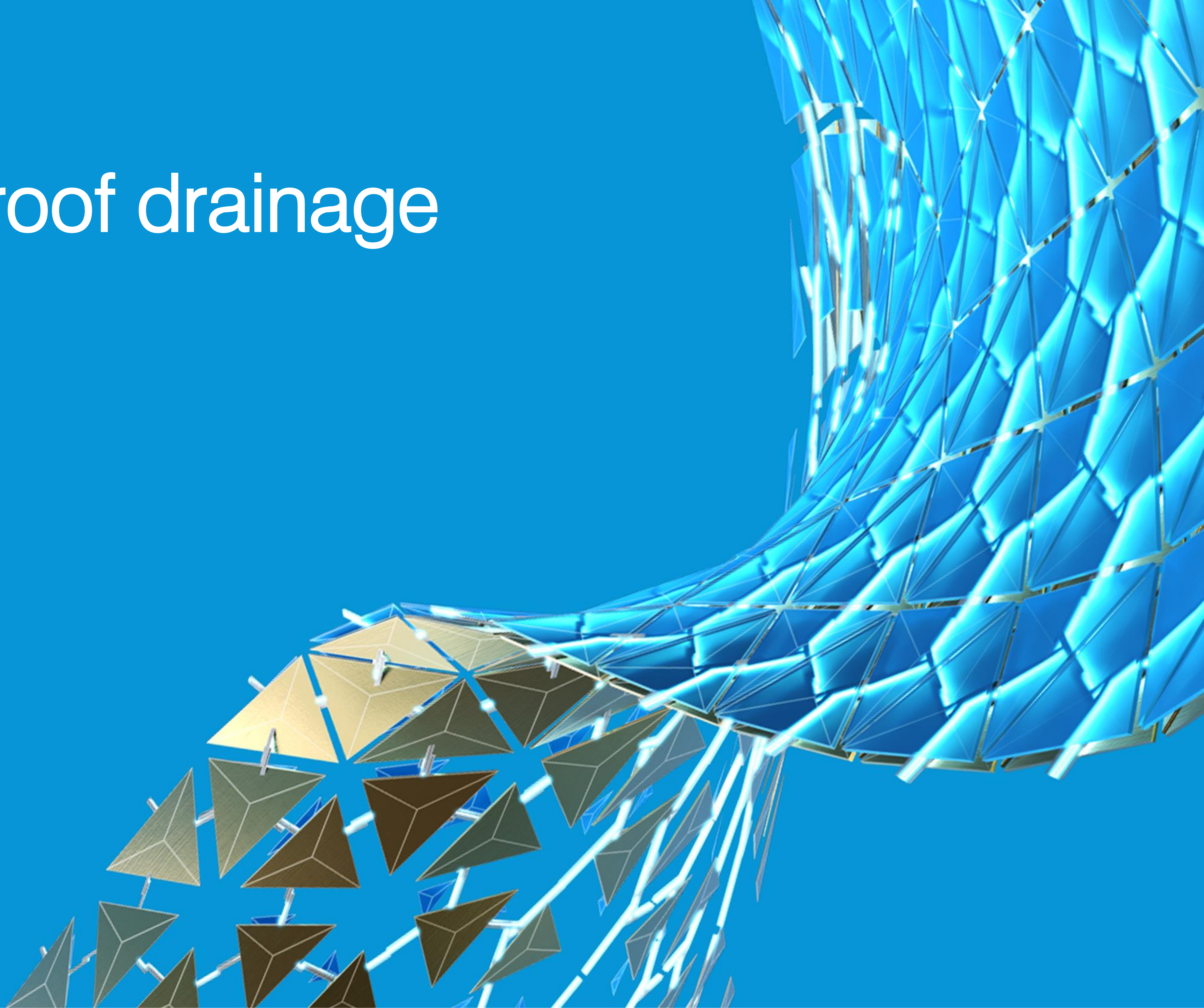
"Estimation of angle of deflection of rain at driving rain prone stations in India, Journal of Asian Architecture and Building Engineering/May 2005/229"

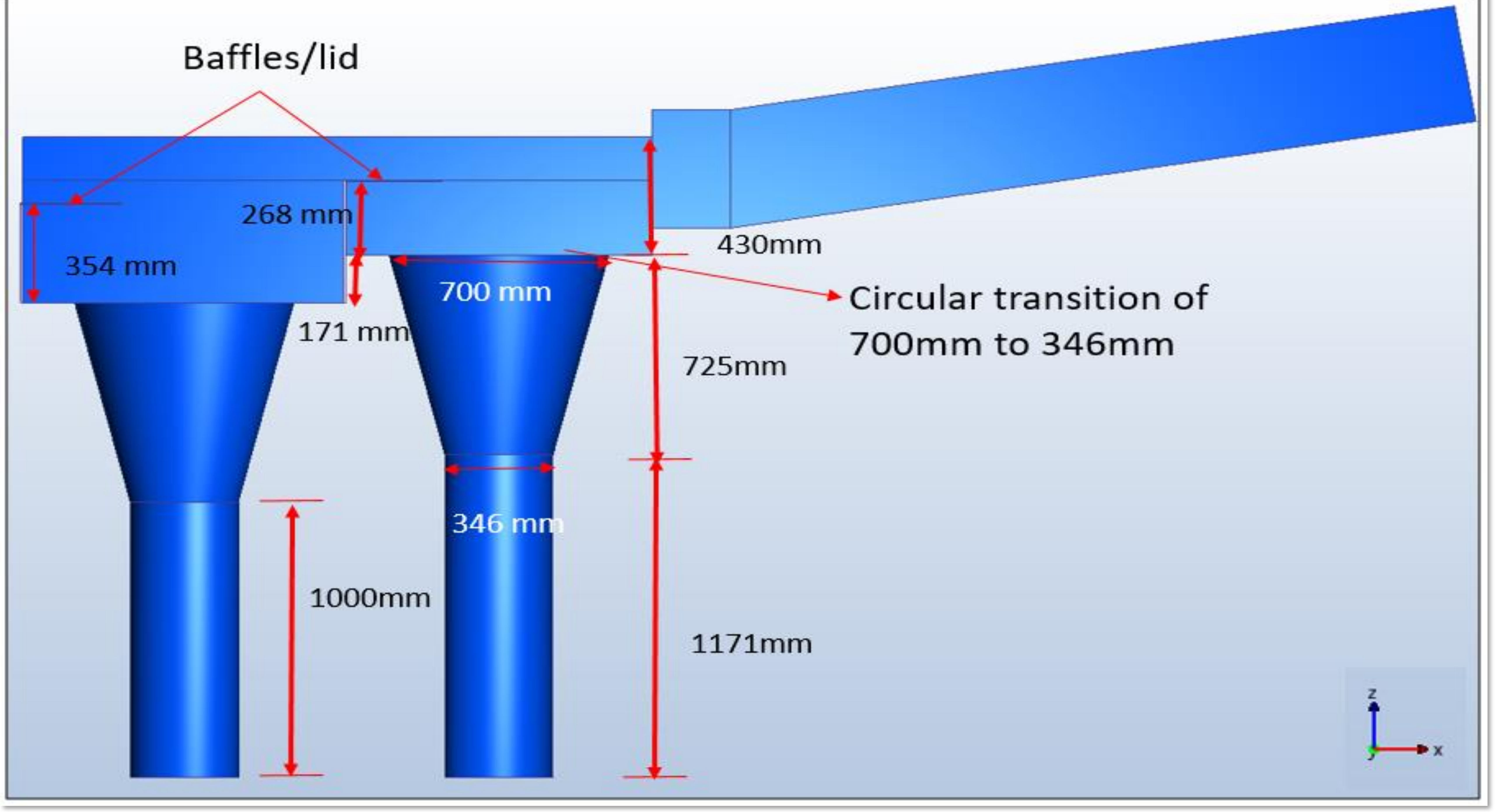
Key outcome

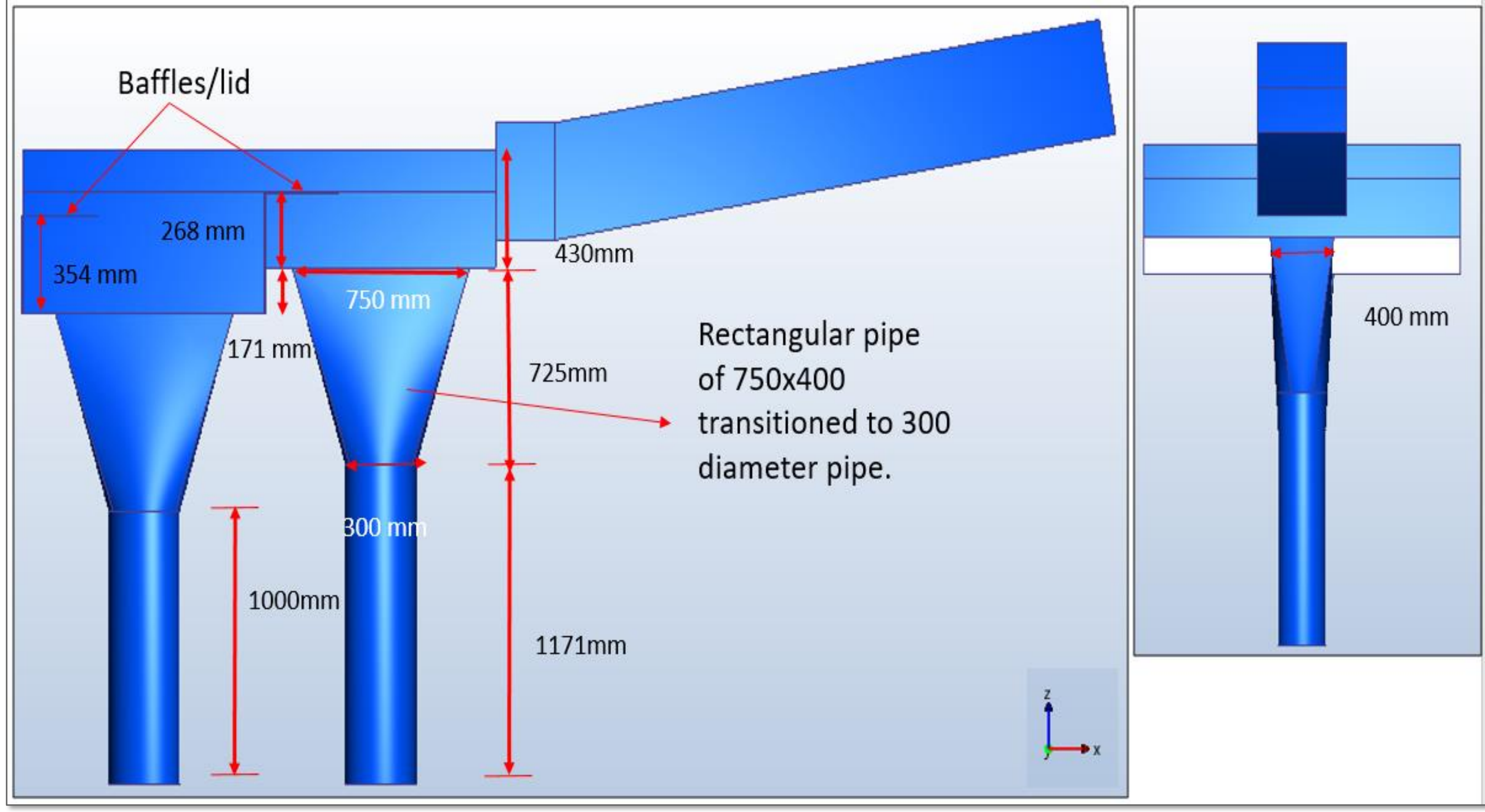
Autodesk CFD Simulation and CFD results provide:

- information on wind flow pattern
- resultant air velocity on the airport roof
- means of evaluating rain deflection angle
- wind driven rainwater ingress into airport forecourt area

Storm water roof drainage







Original design

circular bell mouth pipe of 700mm diameter to 346mm diameter drain pipe

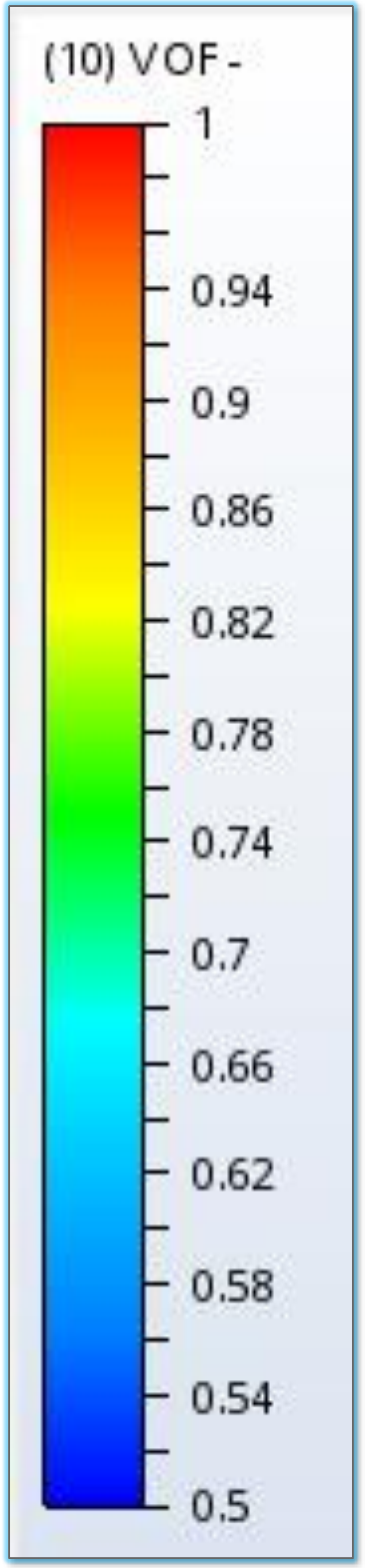
Modified design

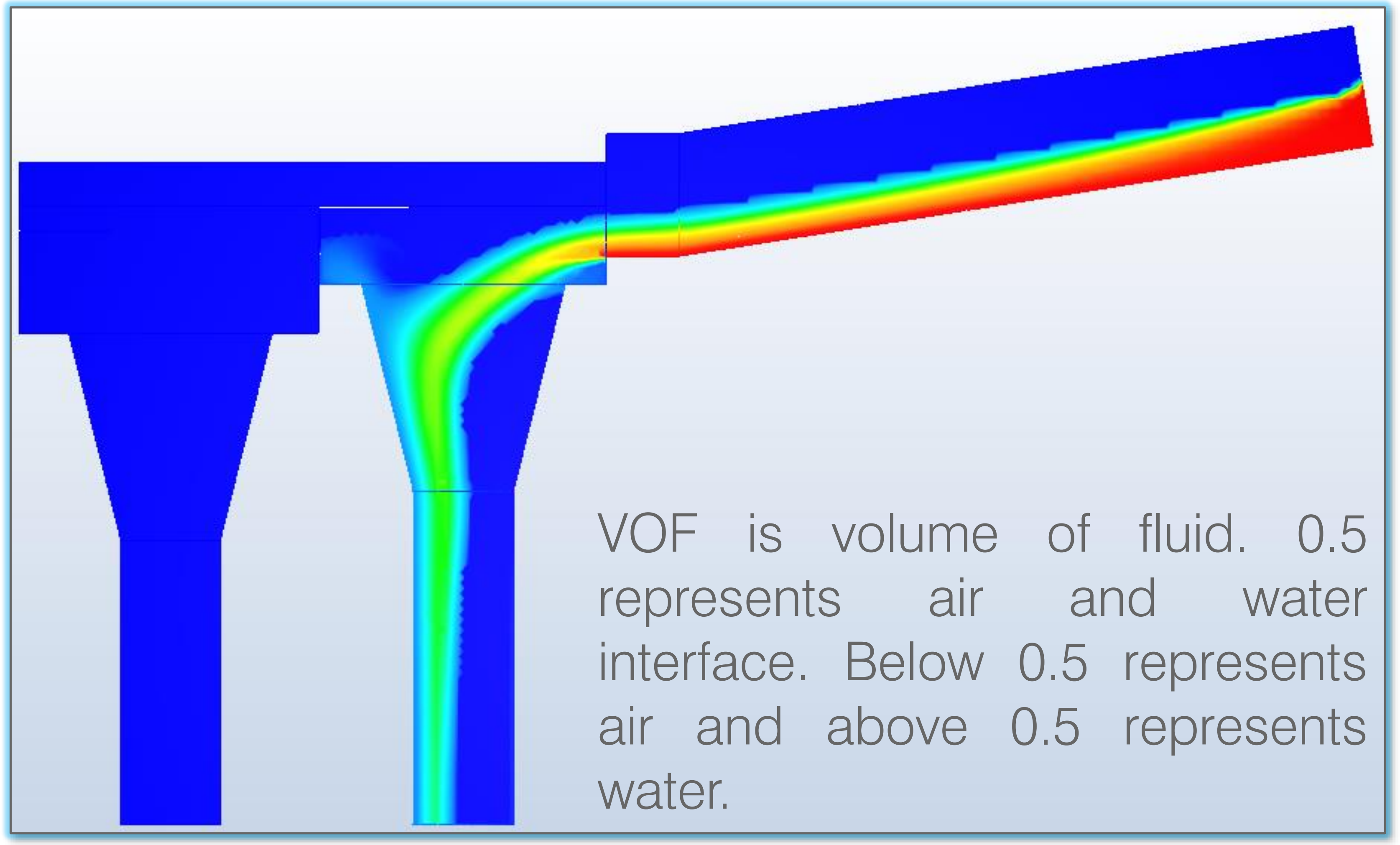
Rectangular bell mouth of 750mm x400mm transitioned to 300mm drain pipe

Rain water flow results

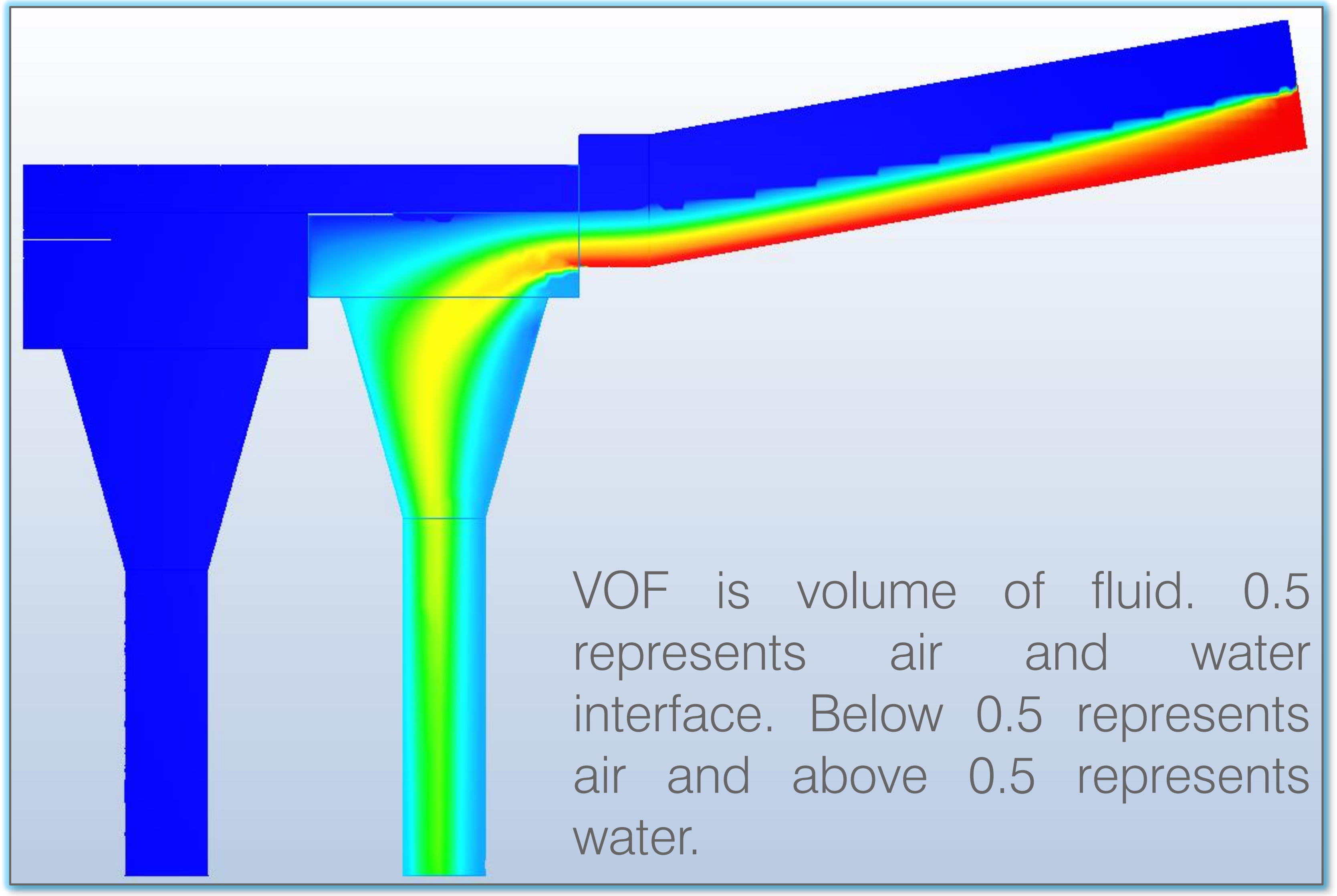
VOF plots in the range of 0.5 to 1.0 for original and modified design

- In the Original design, water flow is not filling the drain pipe cross section
- In the modified design, water fills the drain pipe cross section as it flows down





CFD results of VOF plot for original design

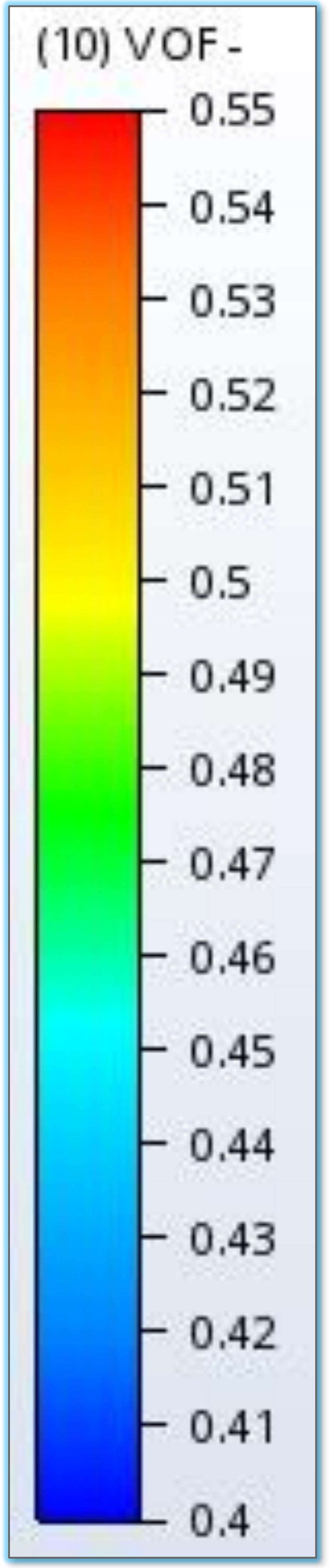


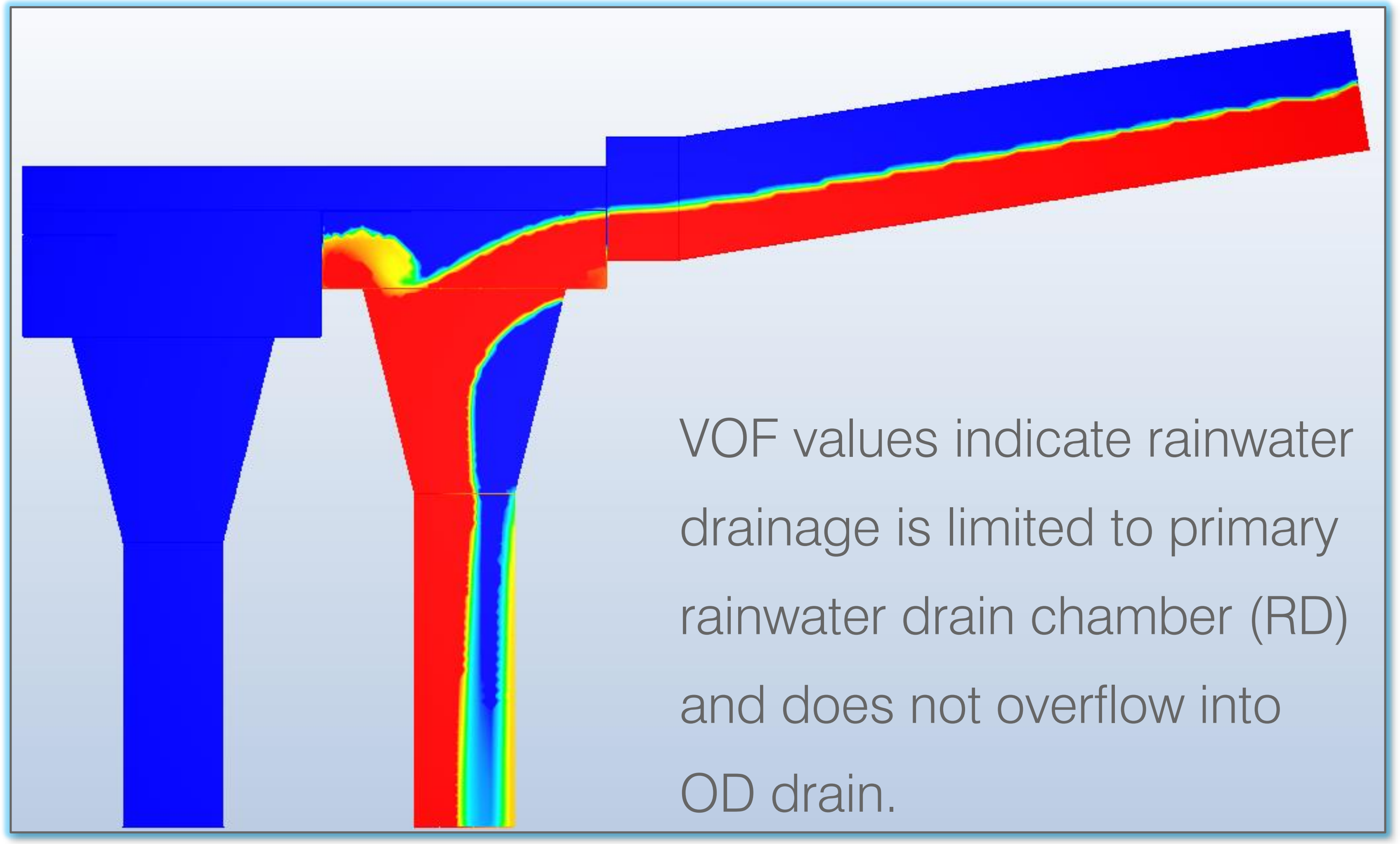
CFD results of VOF plot for modified design

Rain water flow results

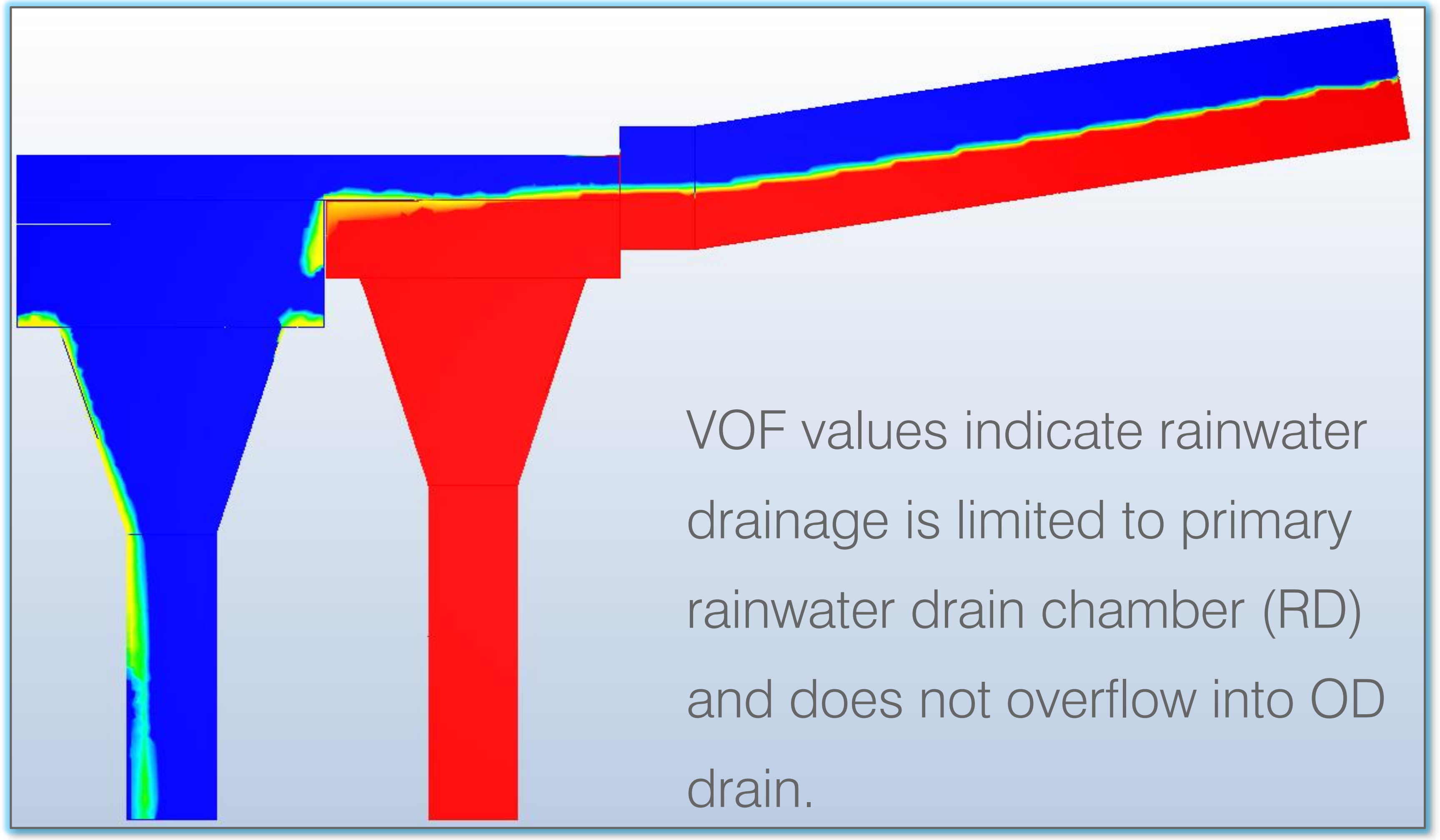
VOF plots in the range of 0.4 to 0.55 for original and modified design

- Rain water in the collection box is not flowing into the overflow drain chamber (OD), entire rain water is flowing through the primary rainwater drain (RD) chamber.
- Rain water flows through the primary rainwater drain chamber (RD) in this option also, with only a minor amount of water spray into the overflow drain chamber (OD).





CFD results of VOF plot for original design



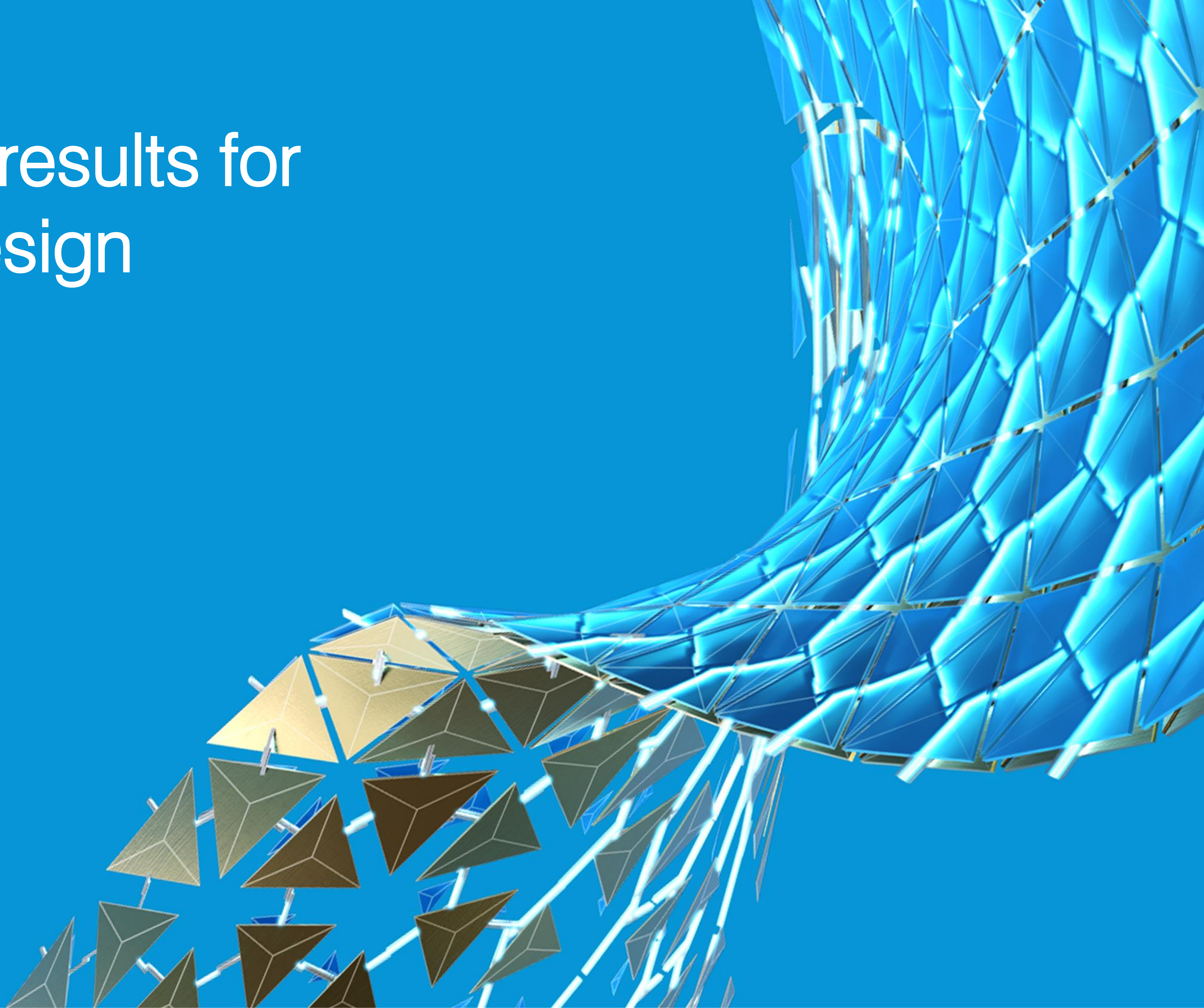
CFD results of VOF plot for modified design

Key outcome

Autodesk CFD was used with free surface flow capability to derive a design that would be feasible to construct at site without impacting existing design of the roof supporting structure, slopes and gutter design.

- Original bell mouth design for roof drainage was found to be not workable since the Structure members supporting the roof assembly did not have the requisite gap to accommodate the transition piece.
- To accommodate the specified bell mouth, major re-design of the roof supporting system was necessary including changing of roof slopes and gutter design.
- Overall impact on design timelines and ultimately on the overall project schedule would have been significant.
- Modified bell mouth design was validated by CFD analysis for its adequacy in rain water drainage for airport roof.
- Through CFD analysis, we demonstrated that this modified design gives acceptable results (same as original design intent) and the same can be integrated with the overall roof design without any modification.

Relevant CFD results for sustainable design



CFD results for sustainable design in AEC

- *Velocity profiles* to understand ventilation effectiveness
- *Temperature profiles* to evaluate thermal comfort in occupied spaces
- *Static pressure distribution* on building façade and roof to understand wind loading
- *VOF (Volume of Fluid)* of water to understand water flow in airport roof rainwater drainage system

Recap

To wrap up, we have seen how Autodesk CFD can be used for sustainable design in AEC industry.

What we have covered in this session for key learnings:

- Air flow and thermal simulation capabilities in Autodesk CFD to address sustainability in AEC design
- Modeling and simulation techniques for energy efficient ventilation design
- Modeling and simulation strategies for wind flow and rain flow on airport building
- Relevant CFD results for sustainable design

Thanks for attending and have a great AU2020!

Pl leave your comments and recommendations on the class page at AU2020

mmrajulu@lntecc.com