468747

# Using Autodesk CFD in AEC Industry for Sustainable Design

Dr. Munirajulu. M
L&T Construction, Larsen & Toubro Limited

## Learning Objectives

- Learn airflow and thermal simulation capabilities in Autodesk CFD
- Discover modeling and simulation techniques for energy efficient ventilation design of occupied spaces
- Learn strategies for modeling and simulation of wind flow and rain flow on airport building
- Explore relevant results from CFD for sustainable design

## Description

In this class, we will cover applications of Autodesk CFD Simulation for sustainable design in AEC Industry. You will learn how Autodesk CFD is used in the AEC design process. We will cover case studies of 1. Energy efficient ventilation design for a large stadium sports facility, 2. Indoor thermal comfort using energy efficient displacement ventilation for large convention center foyer occupied by people. 3. Wind loading analysis to simulate wind pressure and resultant structural loading on airport façade and roof, 4. Optimization of storm water roof drainage design using free surface flow. You will learn how air flow and thermal simulations capability of Autodesk CFD are used to demonstrate performance for sustainable design.

## Speaker(s)

Dr. Munirajulu. M, Bachelor of Technology (Hons.) and Ph.D. from IIT, Kharagpur, India, has more than 24 years of industry experience using CFD technology as an analysis tool in the design of HVAC, Automotive, Fluid Handling Equipment, Steam power plant products. He has been with Larsen & Toubro Limited since 2005 and prior to this, he has worked with ABB Limited and Alstom Projects India Limited for about 9 years. Currently he is responsible for performance-based design using CFD analysis in MEP/AEC areas related to commercial buildings and airports in L&T Construction, Larsen & Toubro Limited, Chennai. He has been using Autodesk CFD Simulation software for MEP/AEC applications in areas such as data center cooling, basement car park ventilation, DG room ventilation, rainwater free surface flow for airport roof design, and smoke simulation in buildings in design stage as well as for trouble shooting. He has been a speaker at AU 2017, 2018, 2019 US and AU India 2019.

## Introduction - Sustainable design in AEC Industry

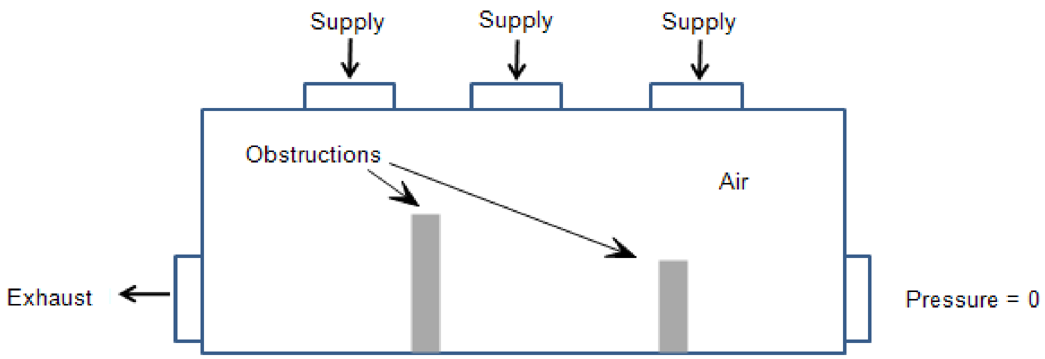
Sustainable practices in design include using digital technologies such as **BIM (Building Information Modeling)** and **Simulation**. This is all the more necessary due to off-site/modular construction and emphasis on use of high-efficiency heating, ventilation and air-conditioning systems. Off-site 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.

## Airflow and thermal simulation capabilities in Autodesk CFD

**Autodesk CFD** is capable of simulating air flow leveraging mechanical ventilation, external flow/wind loading and natural ventilation in **AEC** applications. It can also predict temperature distribution due to heat transfer throughout the building space of interest. Outcome of CFD simulation and analysis are helpful in evaluating design performance for proper airflow and temperature control within the occupied building space. Due to increased use of **BIM (Building Information Modeling)** and **prefab/modular construction** as an attractive alternative to traditional design and construction, CFD simulation helps achieve optimal and improved performance of thermal and ventilation systems, thus promoting sustainable design of buildings. These metrics are defined as follows:

### Mechanical ventilation

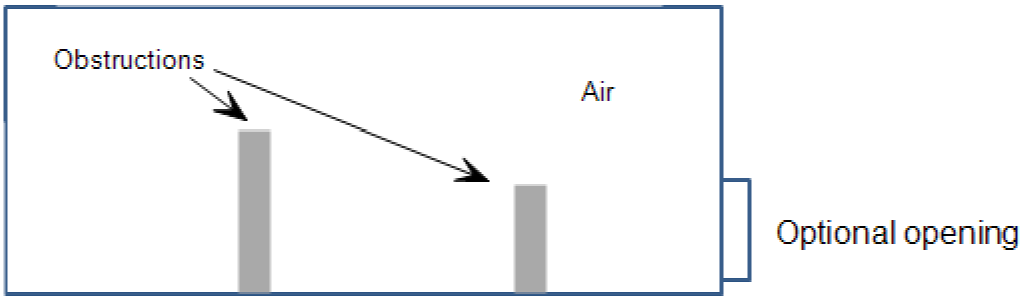
Mechanical ventilation creates *air movement*, driven by fans and provides *air change rates* (the number of times that volume of air in a space is changed per hour) to remove stale air and replace it with fresh air). This results in human comfort and good indoor air quality (removal of contaminants or smoke extraction). In Autodesk CFD, mechanical ventilation analysis deals with *air regions, internal obstructions, supply vents/diffusers/fans and exhaust registers/vents/fans*. Application examples include: *Thermal comfort of occupants within a building, data center cooling analysis, and smoke extraction analysis* for life safety.

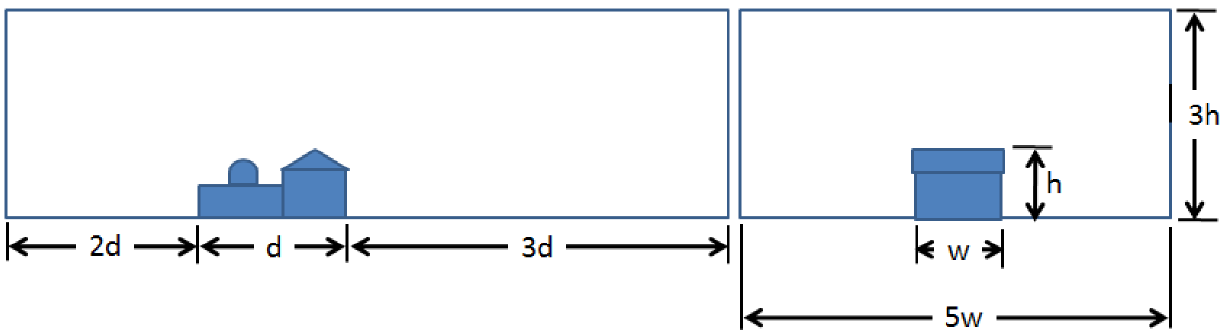



http://help.autodesk.com/view/SCDSE/2019/ENU/?guid=GUID-C52E55AA-38F7-4116-B83A-5AEDF9978F24

### Natural ventilation

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



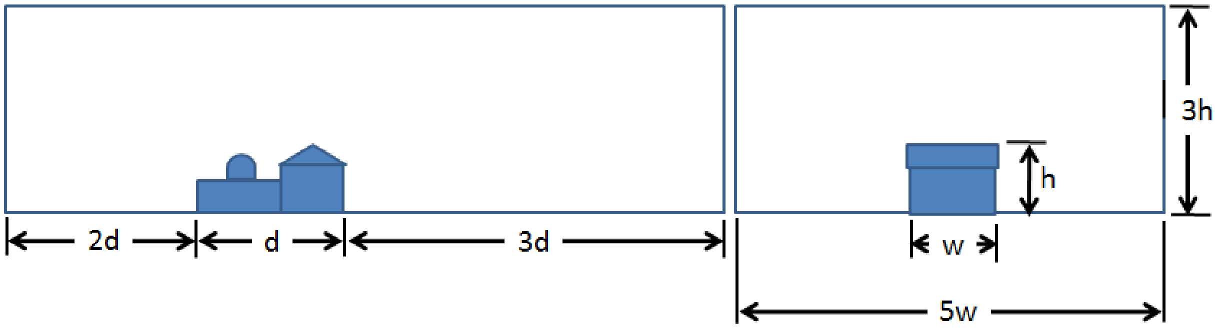




[http://help.autodesk.com/view/SCDSE/2019/ENU/?guid=GUID-F6011744-B69B-47D0-BB2B-50E1BA8DC32D](http://help.autodesk.com/view/SCDSE/2019/ENU/?guid=GUID-F6011744-B69B-47D0-BB2B-50E1BA8DC32D)

### External flow / Wind loading

Autodesk CFD can be used to simulate air flow over and around buildings. Air flow due to wind (*velocity, pressure*) and resulting force on the building structure can be predicted by CFD simulation and informed design decisions can be made based on CFD results. Application examples include: *Flow over and around buildings for ventilation, and wind loading on building facades and/or roofs*.

In Autodesk CFD, to analyze wind effects, surrounding open environment can be created as a volume of air and prescribed wind velocity defined on a face of that volume depending on the local wind direction.




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

### Free surface Modeling

Autodesk CFD can be used to simulate rainwater flow over airport roof and optimize roof drainage design. This can result in material and cost savings.

## Modeling and simulation strategies for energy efficient design of occupied spaces

Indoor thermal comfort and adequate ventilation are important in buildings occupied by people. Modeling and simulation of air flow and heat transfer provides an opportunity to evaluate and optimise design of energy efficient systems used for occupied spaces. In the following project examples, use of Autodesk CFD for ventilation design of (i) a large sports facility and (ii)foyer of a convention center are described.

### (i) Energy Efficient Ventilation Design for a large stadium sports facility

Design of natural ventilation aided by energy efficient fans is a type of sustainable design. Autodesk CFD Simulation can be used to validate the design for acceptable ventilation requirements of a **Sports Training facility- Multi Purpose Hall**. Using *Computational Fluid Dynamics (CFD)* analysis, airflow velocity distribution at various levels in the sports facility is evaluated for acceptable ventilation. Airflow velocity not exceeding **0.15 m/s** in the playing zone (at 4m height) will meet acceptance criterion for ventilation.

From the CAD geometry of multipurpose hall, the model suitable for CFD is prepared and the analysis is carried out using Autodesk CFD for the following scenarios:

- Scenario 1: Model with axial fan design scheme.
- Scenario 2: Model with propeller fan design scheme.
- Scenario 3: Model with modified design for louver size for Scenario 2.

Input details for CFD analysis include building geometry from CAD data, ventilation scheme details (louver size and layout, fan sizes and layout).

## Scenario 1

Figure 1 shows floor plan for sports facility for Scenario1

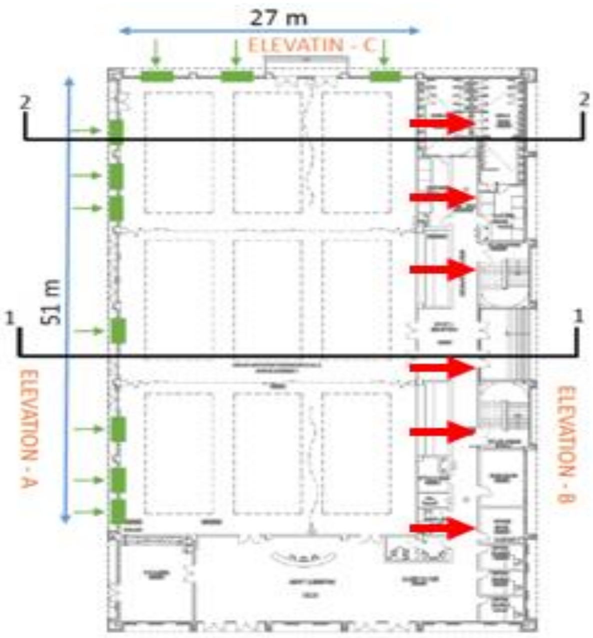



FIGURE 1: FLOOR PLAN FOR AXIAL FAN DESIGN SCHEME – SCENARIO 1

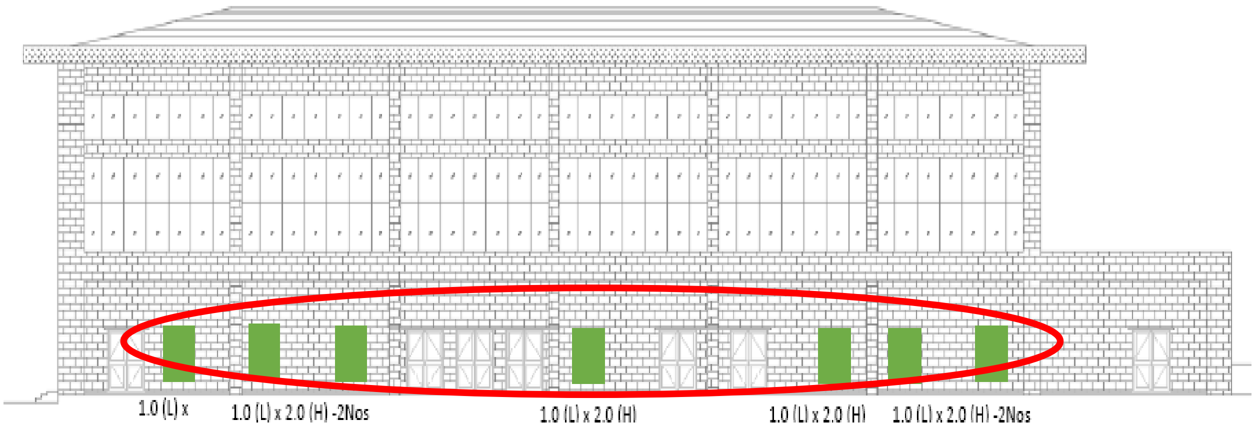



FIGURE 2: ELEVATION A, SIDE VIEW FOR LOUVER SIZE SPECIFICATION OF AXIAL FAN DESIGN SCHEME- SCENARIO 1

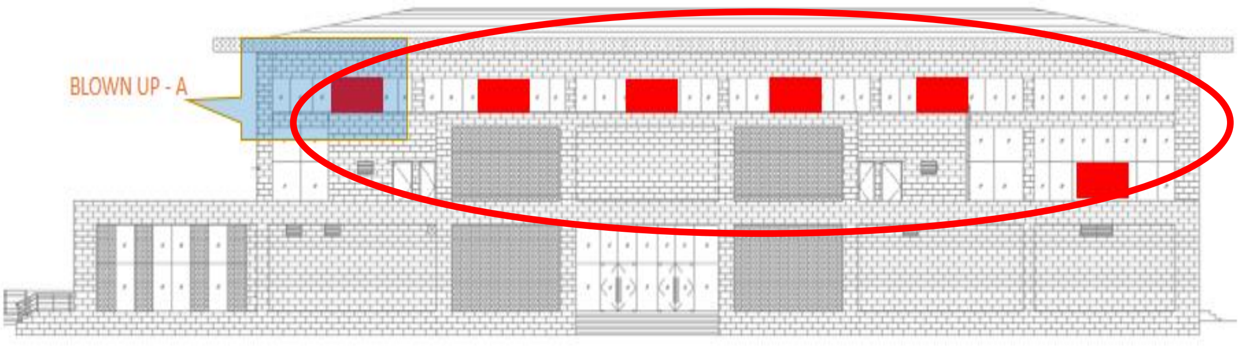



FIGURE 3: ELEVATION B, SIDE VIEW OF AXIAL FAN SCHEME- SCENARIO 1

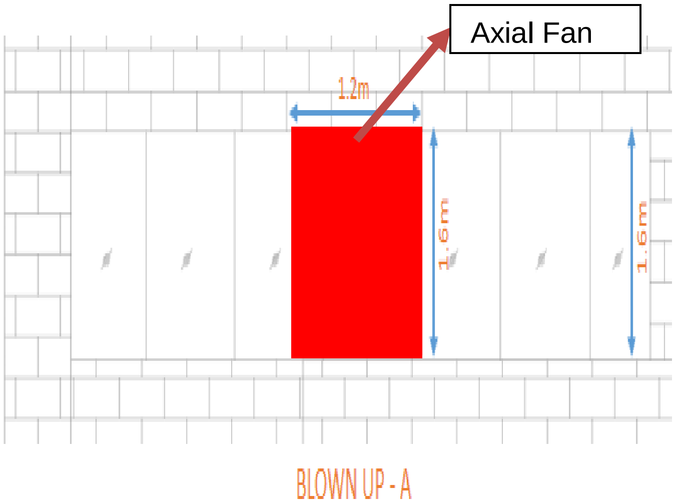



FIGURE 4: IMENSIONAL DETAILS FOR AXIAL FAN SCHEME – SCENARIO 1

Ventilation design scheme for **Scenario 1** includes seven louvers on elevation side A as shown in Figure 1. There are three louvers on elevation side C as shown in Figure 1. Six axial fans are kept on elevation side B as in Figure 3. Each axial fan has a capacity of 10094 CFM. The louvers have a free area ratio of 0.5

Geometry of the multi-purpose hall is simplified by removing detail that is not relevant to the simulation e.g. louver casings and axial fans, as shown in Figure 5, 6 and 8. Complicated components such as axial fans are replaced with simple cuboid. Airtight CAD model is prepared so that the model contains an air volume within the building space. To ensure adequate spacing between the inlet openings and neighboring walls, geometry is extended at least a hydraulic diameter of the opening. This ensures enough space between the *Boundary Conditions (BCs)* and the internal flow, as shown in Figure 7 and 9.

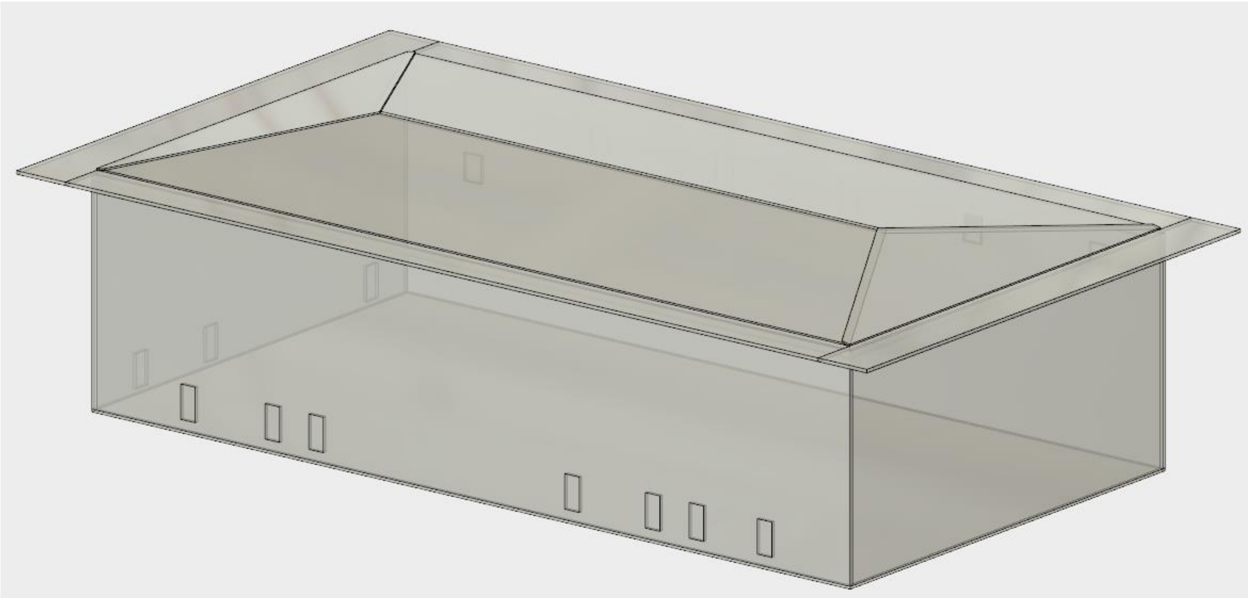


FIGURE 5: CAD MODEL FOR CFD SIMULATION – SCENARIO 1

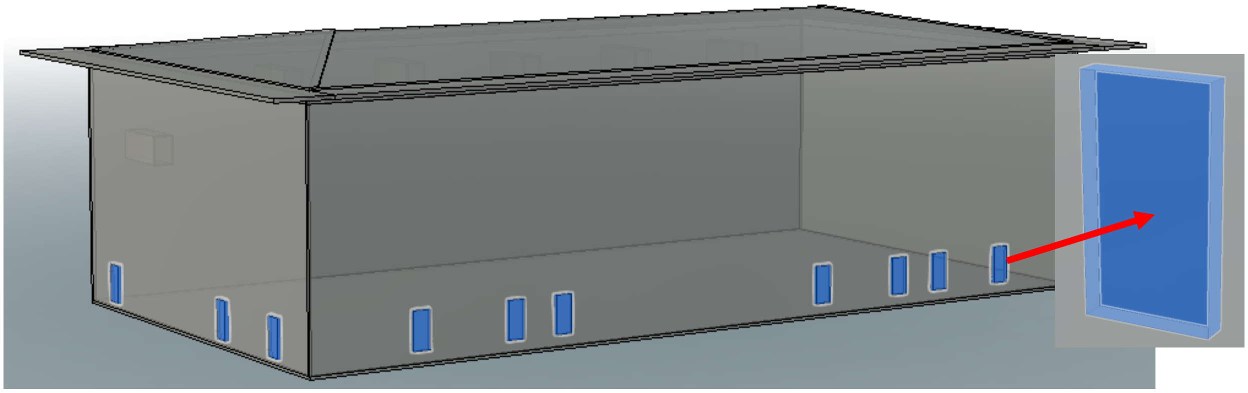


FIGURE 6: AIR INLET LOUVERS AND INLET LOUVER MODELED AS CAD PART – SCENARIO 1

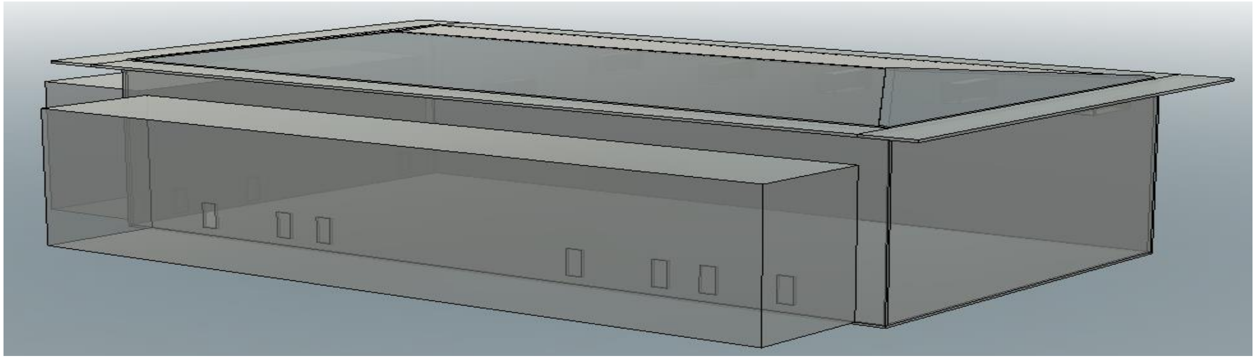


FIGURE 7: INLET EXTENSION MODELED AS CAD PART – SCENARIO 1

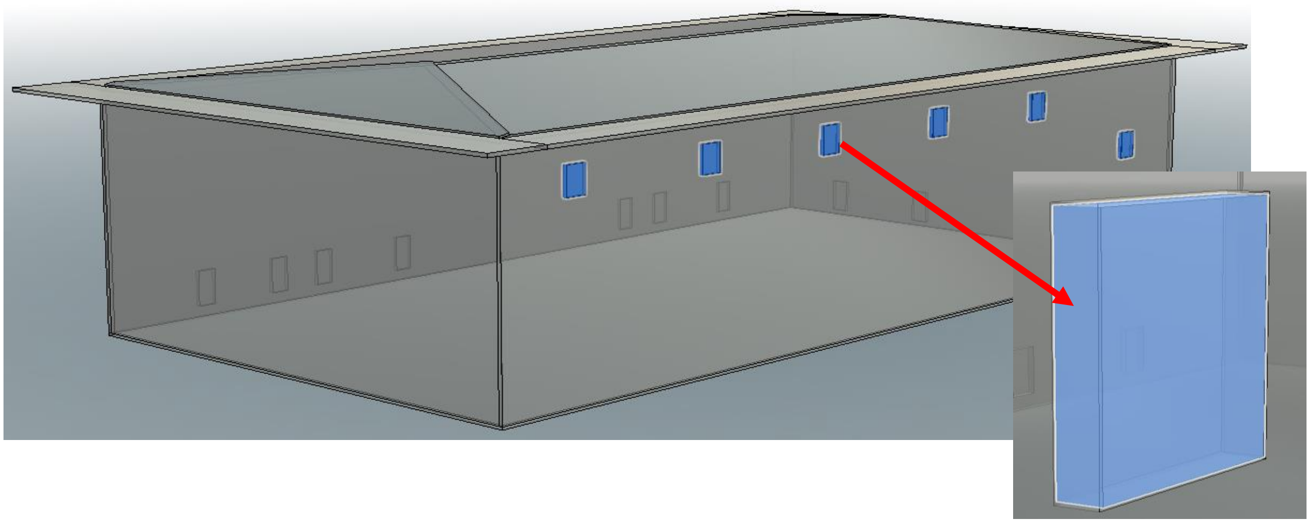


FIGURE 8: AXIAL FAN DETAILS AND AXIAL FAN MODEL AS CAD PART – SCENARIO 1

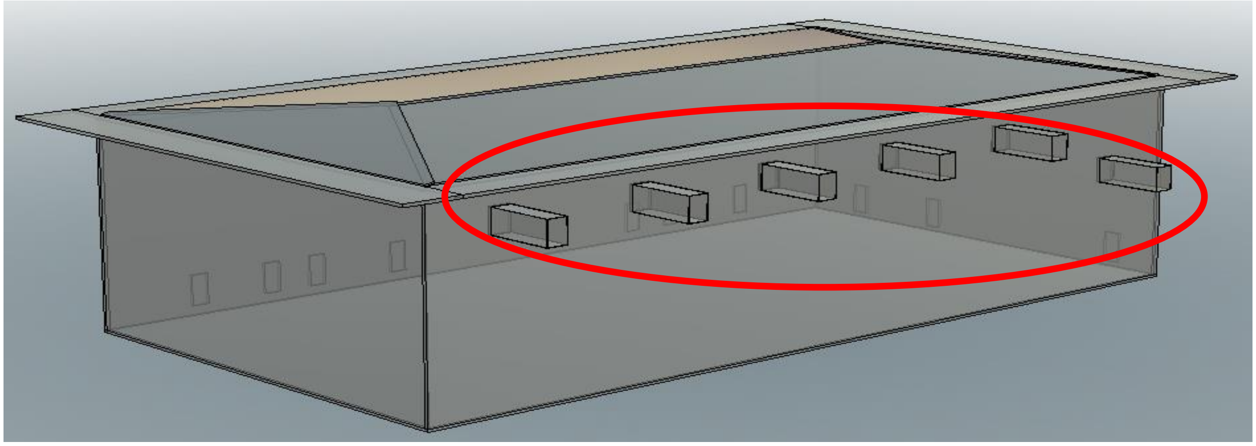


FIGURE 9: OUTLET EXTENSION MODELED AS CAD PART – SCENARIO 1

"*Air*" material is assigned to all fluid regions. The properties of air do not change for mechanical ventilation flows, so **buoyancy effects** are negligible and hence material property environment setting is set to "*Fixed*". Other materials used include solids for building enclosure and resistance for louvers. "*Resistance*" material for louvers is specified with appropriate free area ratio.

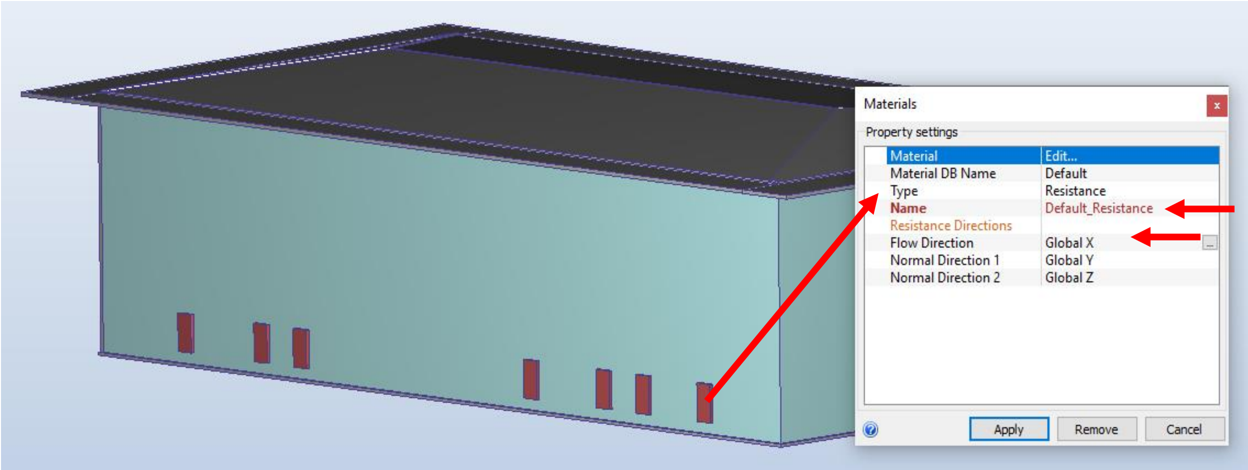



FIGURE 10: CFD MODEL IN AUTODESK CFD WITH FLUID DOMAIN AS "AIR" MATERIAL AND INLET LOUVER AS "RESISTANCE" MATERIAL – SCENARIO 1

*Air* is mechanically moved out of the building space, so outlets are assigned *volume flow rate* on each axial fan outlet, (see Figure 11). Arrow direction indicates the flow in the right direction from the axial fan outlet. *Pressure* boundary condition is applied on the inlet surface of the air volume extension on the louvers, (see Figure 12).

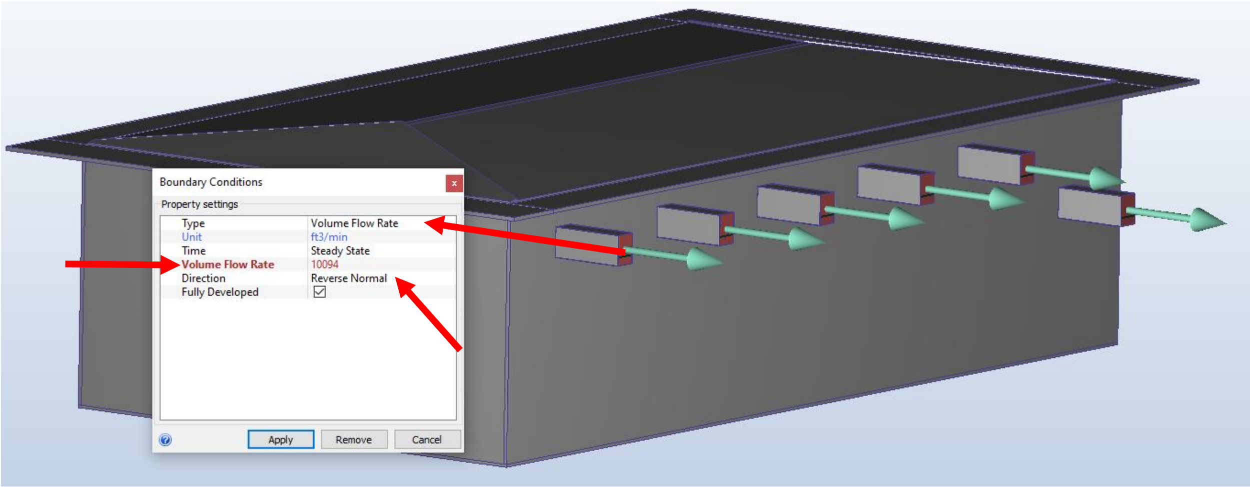



FIGURE 11: CFD MODEL IN AUTODESK CFD WITH OUTLET BC (AXIAL FAN FLOW CAPACITY)- SCENARIO 1

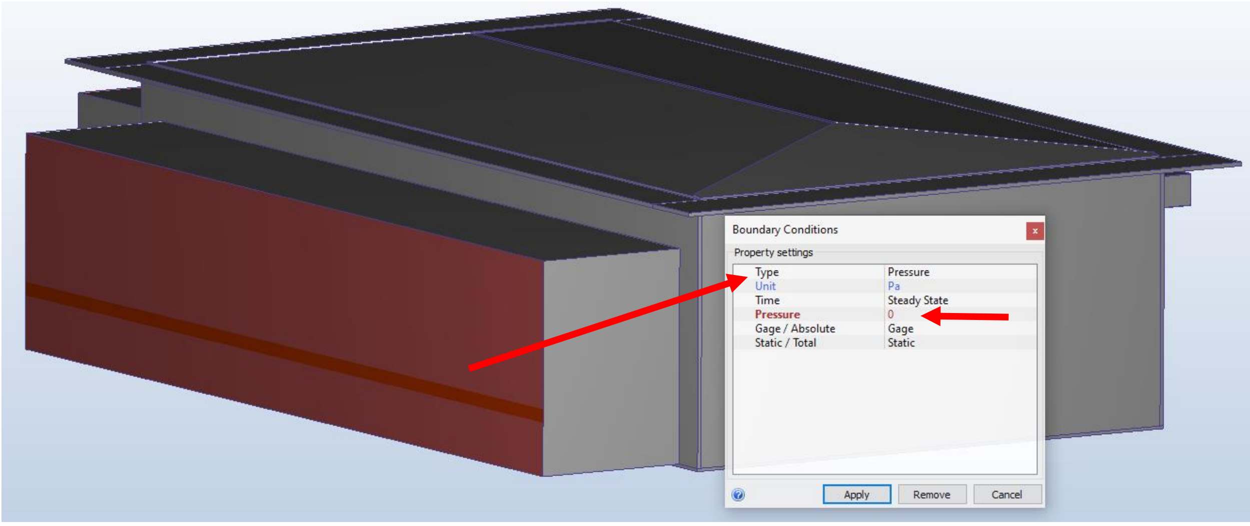



FIGURE 12: CFD MODEL IN AUTODESK CFD WITH INLET BC ON INLET EXTENSION – SCENARIO 1

*Automatic mesh sizing* is used to define the mesh distribution in the CFD model domain. *Local mesh refinement* is used for mesh on louvers and axial fan outlets to capture the flow through them, as shown in Figure 13.

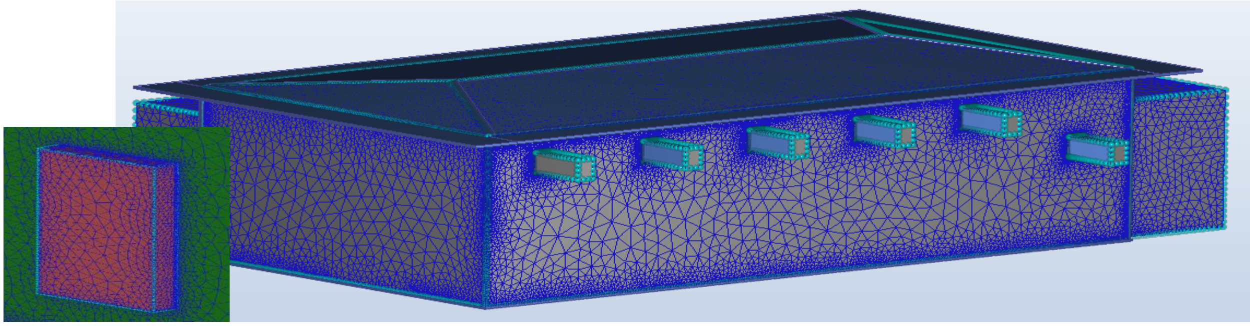


FIGURE 13: CFD MODEL IN AUTODESK CFD WITH AUTOMESH FOR THE VOLUME AND REFINED MESH FOR INLET LOUVER – SCENARIO 1

Flow is solved with *incompressible*, *steady state* flow option. There is no *heat transfer* considered in this analysis so heat transfer is not enabled in the solution settings, as shown in Figure 14.

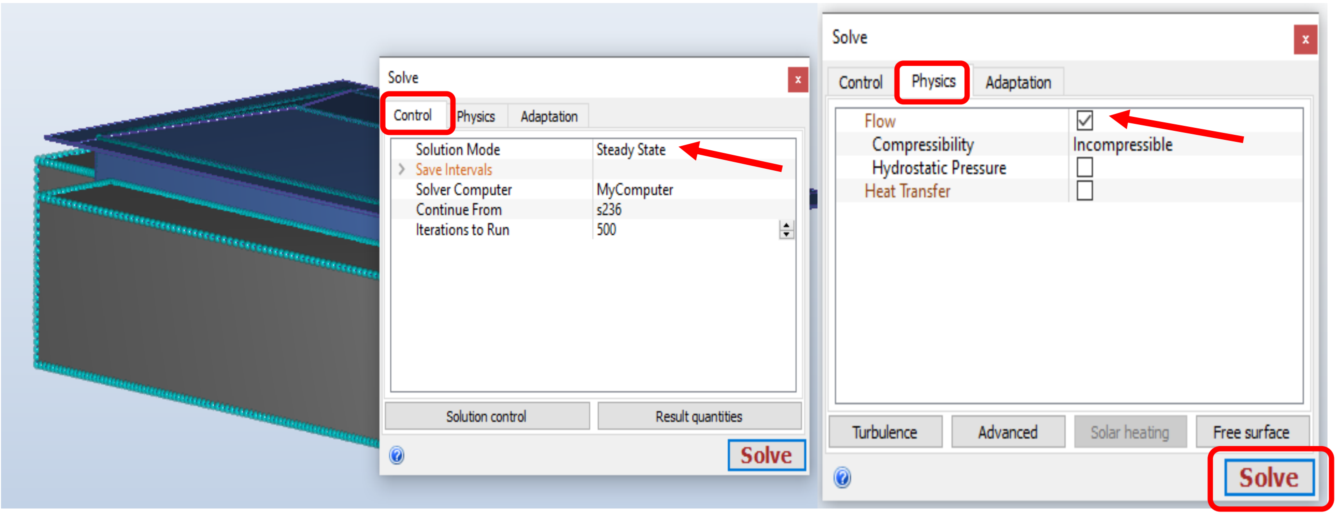
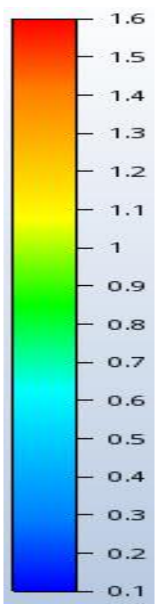



FIGURE 14: STEADY STATE AND INCOMPRESSIBLE FLOW PHYSICS – SCENARIO 1

*Results Planes* are used to visualize the air flow within the multi-purpose hall, as shown in Figure 15 and 16. *Velocity* plots at 1m and 4m level from the floor are extracted from CFD simulation to evaluate the effectiveness of ventilation.




Velocity m/s

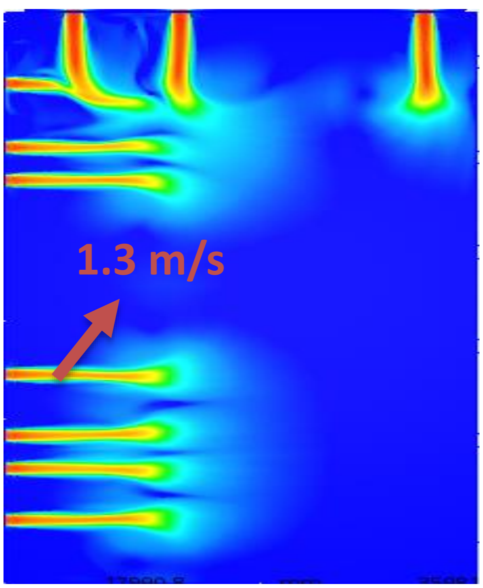



FIGURE 15: VELOCITY PROFILE AT 1M FROM FLOOR

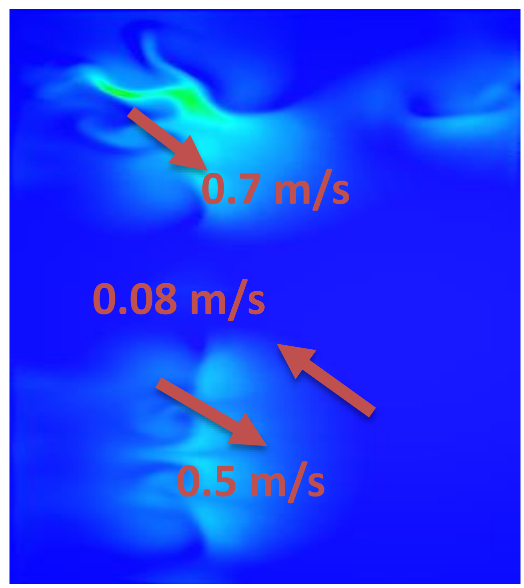



FIGURE 16: VELOCITY PROFILE AT 4M FROM FLOOR

### *CONCLUSIONS - Scenario 1*

CFD results at a height of 1m shows streams of high velocity airflow coming out from the louver region into playing zone, as shown in Figure 15. CFD results at a height of 4m show that the velocity values are higher than the recommended value of **0.15 m/s**, as shown in Figure 16. Therefore, this design does not meet acceptance criteria.

### Scenario 2

**Scenario 2** design consists of same louver scheme on elevation A and C as in **Scenario 1**, as shown in Figure 1. There are eighteen *propeller fans* on elevation B, as shown in Figure 17. Each propeller fan has a capacity of 3364 CFM. The louvers have a free area ratio of 0.5. CFD modeling strategy for Scenario 2 is the same as Scenario 1.

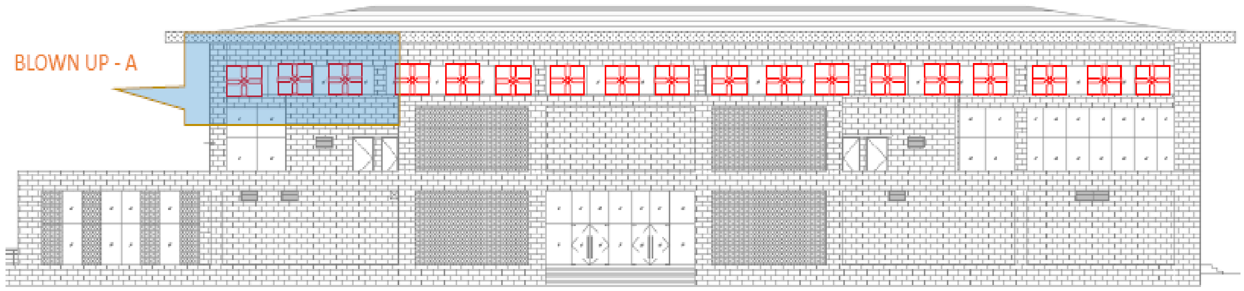



FIGURE 17: ELEVATION B SIDE VIEW OF PROPELLER FAN SCHEME – SCENARIO 2

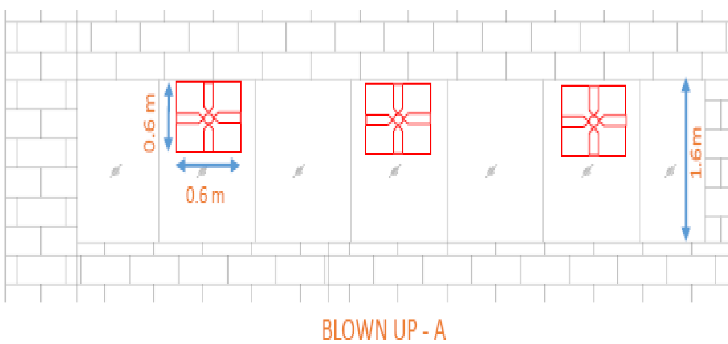



FIGURE 18: DIMENSIONAL DETAILS FOR PROPELLER FAN SCHEME

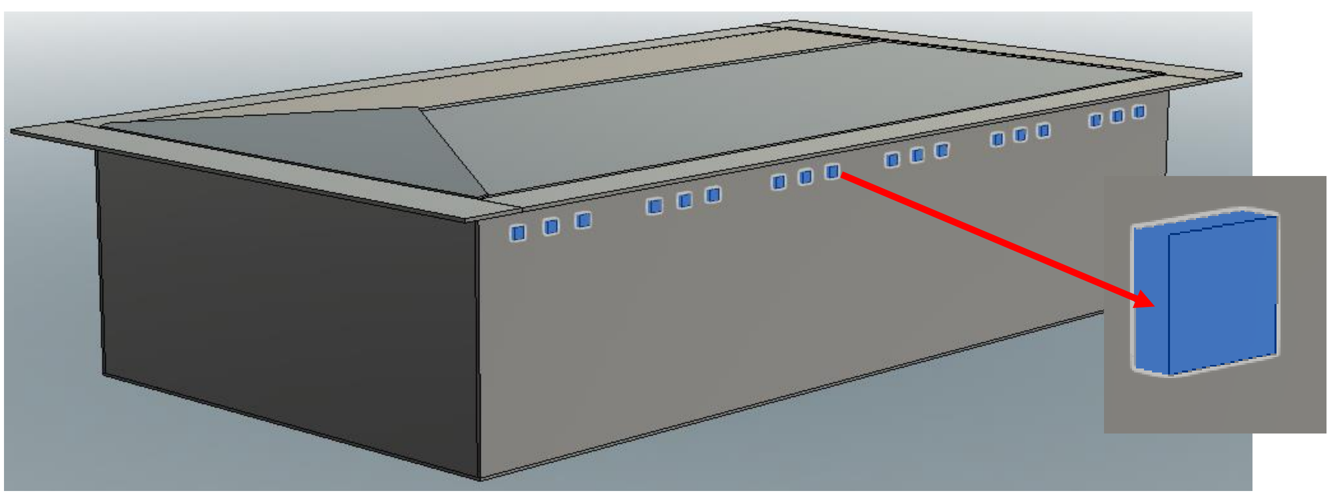


FIGURE 19: PROPELLER FAN DETAILS AND PROPELLER FAN MODEL AS CAD PART – SCENARIO 2

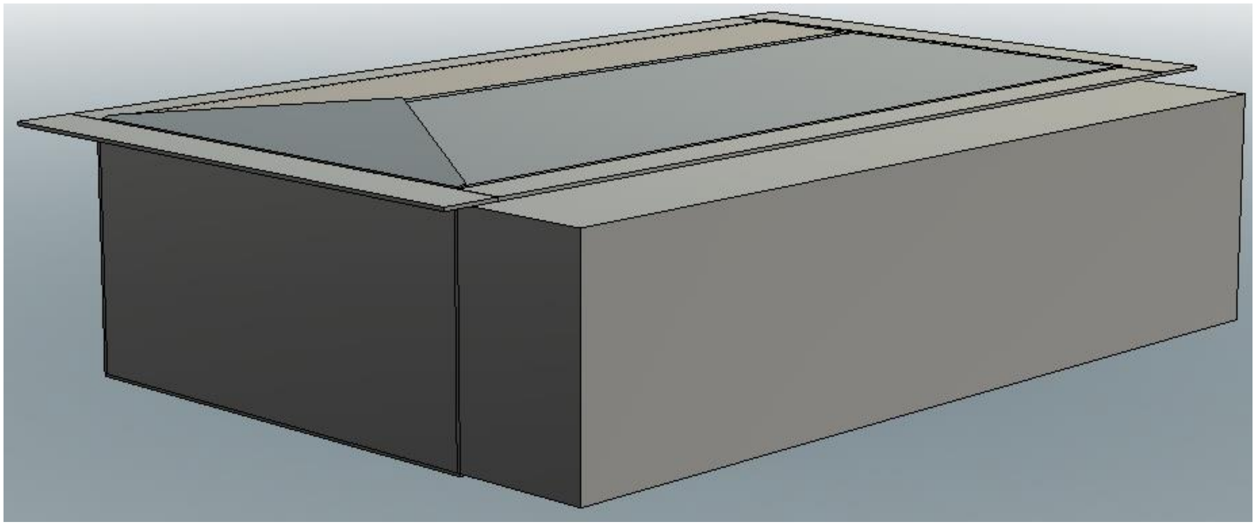
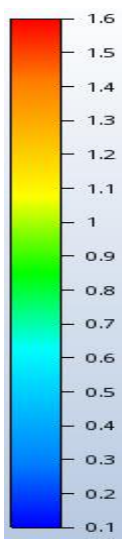


FIGURE 20: OUTLET EXTENSION MODELED AS CAD PART – SCENARIO 2




Velocity m/s

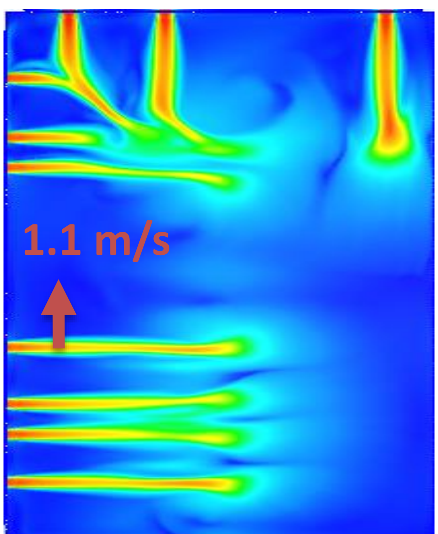



FIGURE 21: VELOCITY PROFILE AT LEVEL 1M FROM FLOOR

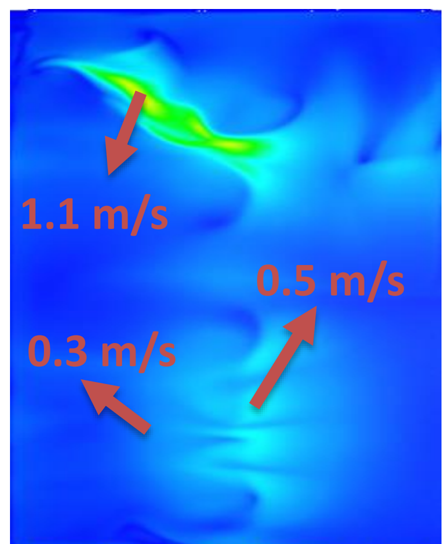



FIGURE 22: VELOCITY PROFILE AT LEVEL 4M FROM FLOOR

***CONCLUSIONS - Scenario 2***

CFD results at 1m height show similar airflow pattern as in Scenario 1 with elongated velocity stream. CFD results at a height of 4m from the floor also show that airflow velocity values exceed **0.15 m/s** which does not meet acceptance criterion.

## Scenario 3

Design scheme for **Scenario 3** is based on the outcome of **Scenario 2** analysis. In **Scenario 3**, louver size is modified from (1.0 x 2.0) m to (1.2 x 2.0) m with elevation C louvers turned off and the propeller fans are set to a ventilation rate of 4 **ACH** (Air Changes per Hour).

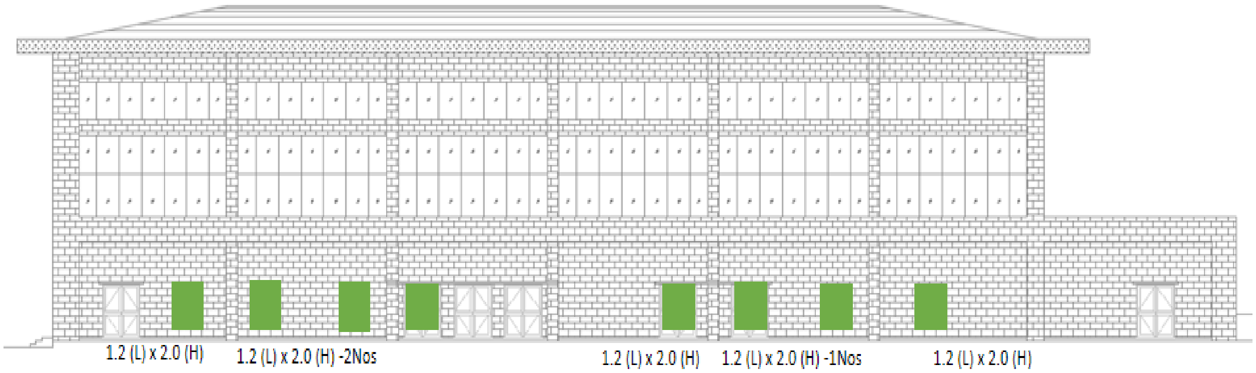



FIGURE 23: ELEVATION A SIDE VIEW FOR LOUVER SIZE SPECIFICATION OF SCENARIO 3

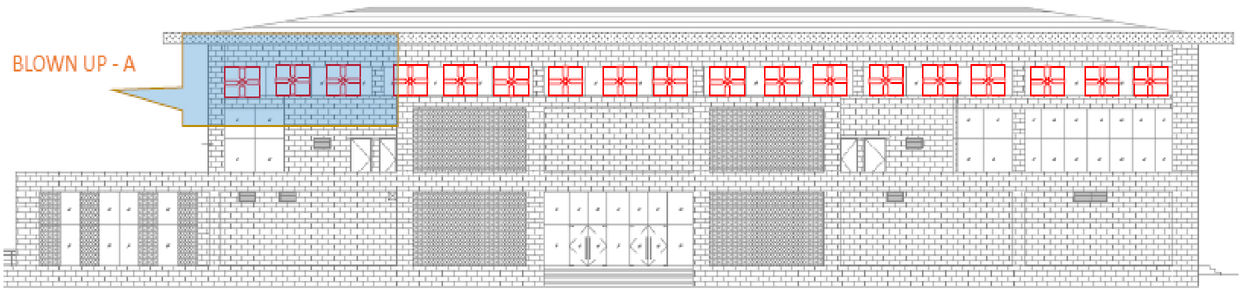



FIGURE 24: ELEVATION B, SIDE VIEW OF PROPELLER FAN SCHEME FOR SCENARIO 3

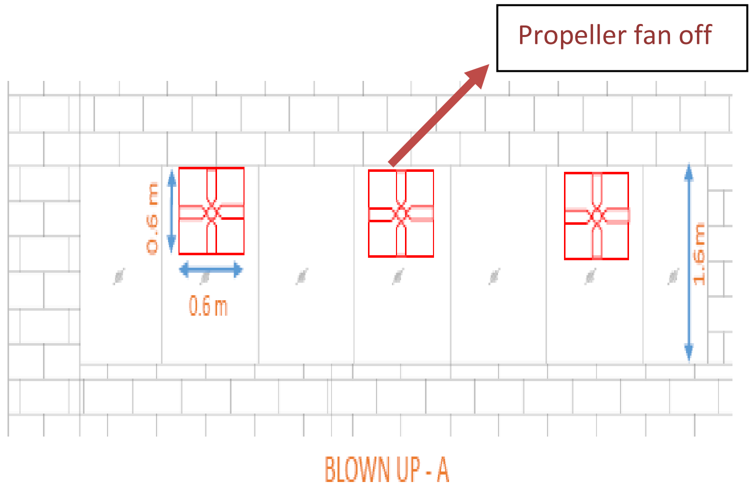
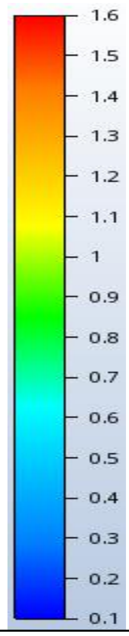



FIGURE 25: DIMENSIONAL DETAILS FOR PROPELLER SCHEME

**Scenario 3** scheme consists of eight louvers on elevation side A, as shown in Figure 23. Three louvers on elevation side C, (see Figure 1) are turned off. This scheme consists of eighteen propeller fans for exhaust, as shown in Figure 24. Some propeller fans are turned off to provide 4 **ACH** ventilation rate. Each propeller fan has a capacity of 3364 cubic feet per minute. The louvers have a free area ratio of 0.5. CFD modeling strategy for Scenario 3 is the same as Scenario 1.




Velocity m/s

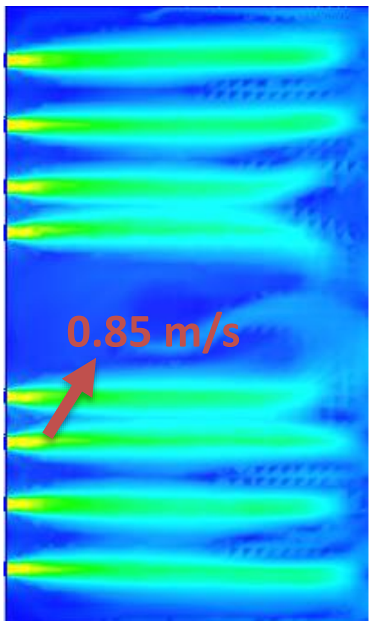



FIGURE 26: VELOCITY PROFILE AT ELEVATION 1M FROM FLOOR

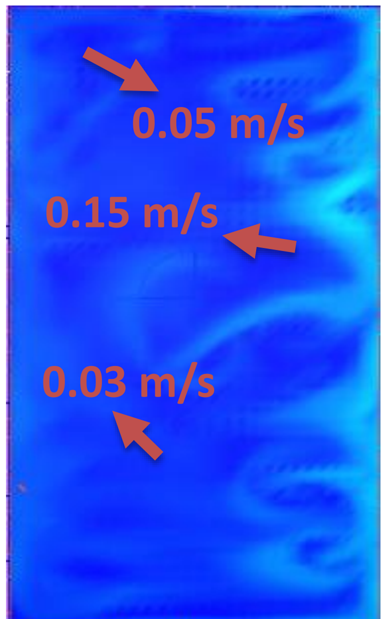



FIGURE 27: VELOCITY PROFILE AT ELEVATION 4M FROM FLOOR

### *CONCLUSIONS - Scenario 3*

CFD results show that the airflow velocity profile at level of 1m from the floor is streamlined without any recirculation zone. The results also show that the airflow velocity at height of 4m from the floor does not exceed **0.15 m/s** in the playing zone region. The airflow velocity values arrived in this case are within acceptable limit of **0.15 m/s** and hence *<u>this design is acceptable and recommended</u>*.

### Energy savings

Electric power required for six axial fans of 10094 CFM, as in **Scenario 1**, is 9 KW while the power required for eighteen propeller fans, as in **Scenario 3**, is 6.7 KW. Therefore, **Scenario 3** with propeller fan option is *energy efficient* as it leads to *energy savings* of about 26%. Also **Scenario 3** with propeller fan option results in better and acceptable ventilation for activities related to **Sports Training Facility Multi-Purpose Hall**.

*Natural ventilation aided by energy efficient fans also provides a cost-effective alternative to more energy consuming and costly air-conditioning system.*

***References***

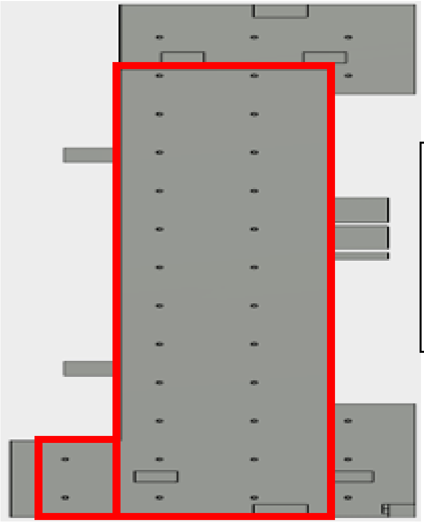
*Determining the flow rates required and temperature of inlet cold stream to provide thermal comfort in badminton court of sama sports complex using CFD – Shah Darshakkumar, Darshan Gajdhar, Devendra Patel.

## (ii) Indoor thermal comfort using energy efficient displacement ventilation for large convention center foyer

*Performance-based HVAC design* is implemented for achieving critical approvals and value engineering using *Displacement Ventilation (DV)* in Foyer of a large convention center. Foyer is a large public space and location of air terminals is driven by architectural/ structural requirements. Return air terminal location is also restricted by architecture/structure. There is no AHU room at the same level and so all the equipment is placed in basement one level below. Challenges imposed by these restrictions are addressed by performance-based CFD analysis to provide an energy efficient and guaranteed solution.

Displacement ventilation (DV) has been found to be useful in large buildings, such as convention center halls, auditorium, and other large height areas. In this type of air-conditioning system, cool air from the diffusers are supplied at low velocities and this dense cool air rises after absorbing heat from occupants, equipment and lighting. This leads to thermally stratified indoor environment. Heated air rises upwards and is discharged through exhausts placed at higher levels. Autodesk CFD Simulation can be used effectively in the design and evaluation of displacement ventilation system which is cost effective for maintaining thermal comfort and reduced cooling costs compared to traditional *mixing air-conditioning* systems, thus providing benefits of sustainable design.


FIGURE 28: FOYER AREA OF CONVENTION CENTER BUILDING



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

FIGURE 29: PLAN VIEW OF CAD MODEL OF FOYER WITH MARKED AREA FOR HEAT LOAD DISTRIBUTION

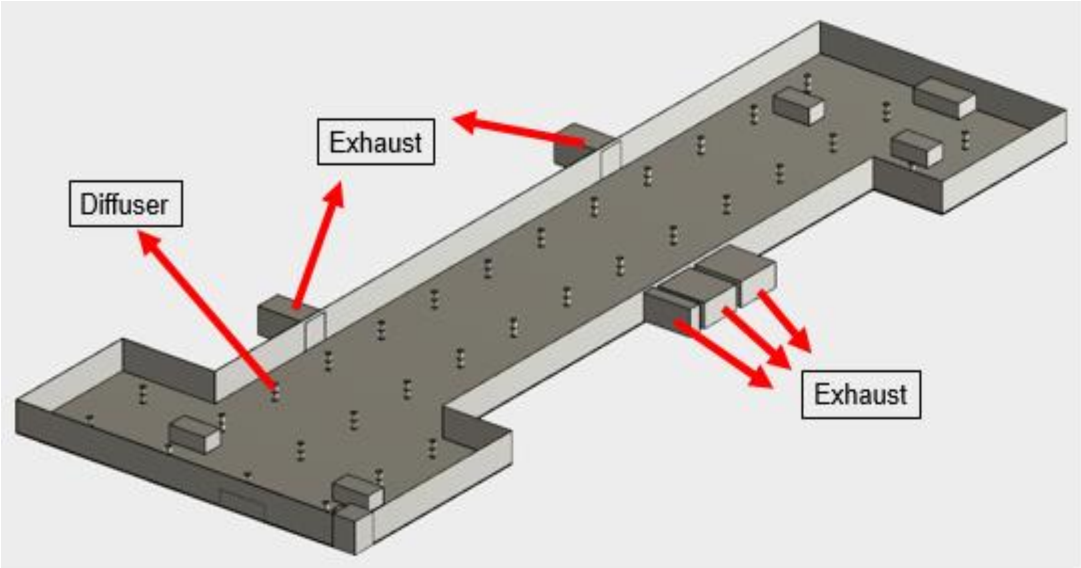



FIGURE 30: CFD MODEL WITH SUPPLY AND RETURN DETAILS FOR AIR-CONDITIONING

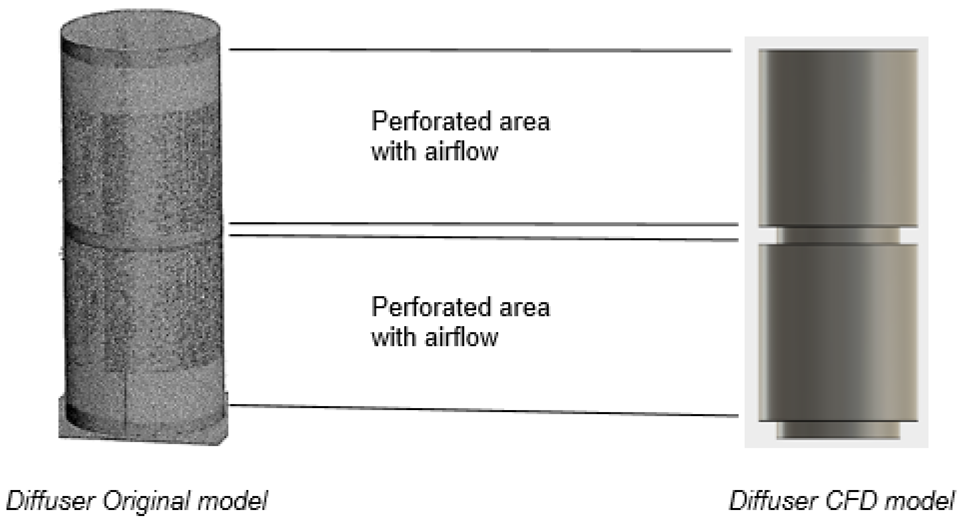



FIGURE 31: COMPARISON OF PHYSICAL MODEL OF DIFFUSER WITH CFD MODEL

Displacement tower diffuser produces a low velocity radial airflow pattern from perforated cylindrical discharge, low-level lake of high quality supply air that floods the floor with airflow. Energy saving also accrues from 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

- 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 +-1 $^{0}$C
- Total heat load = Human load + Equipment load= (5160 * 71) W + (82000) W= =446230 W

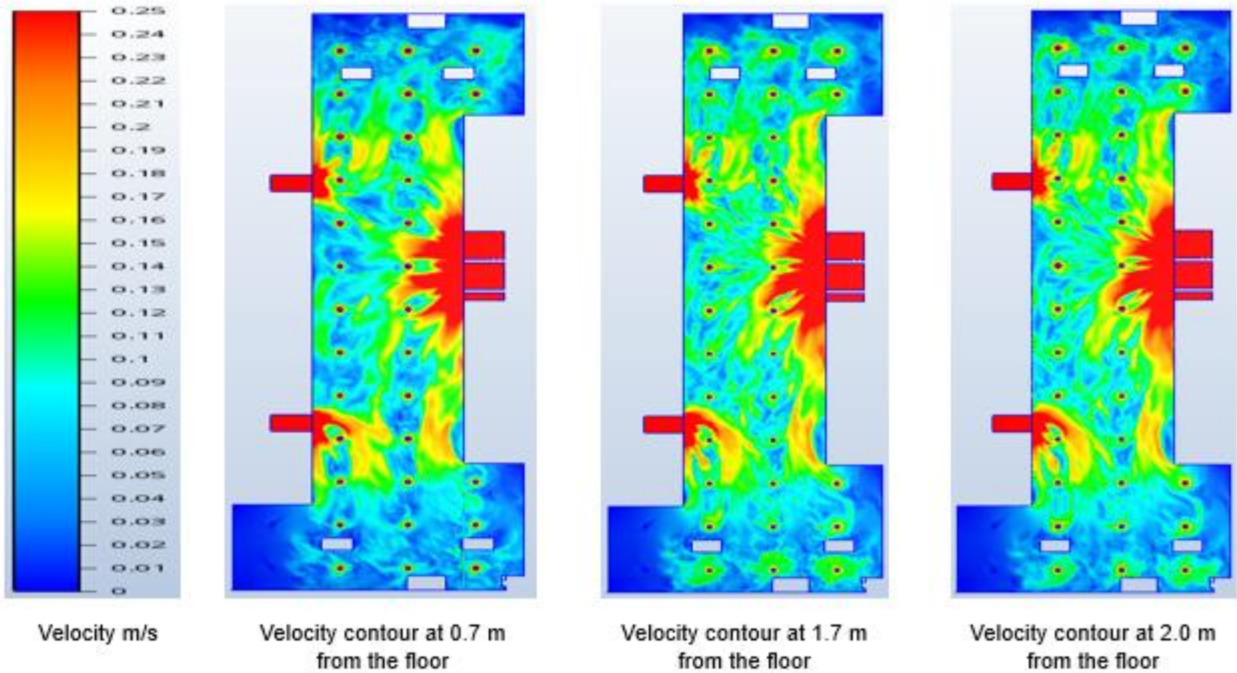



FIGURE 32: AIR FLOW DISTRIBUTION (RESULTS FROM AUTODESK CFD USING PLANES OPTION)

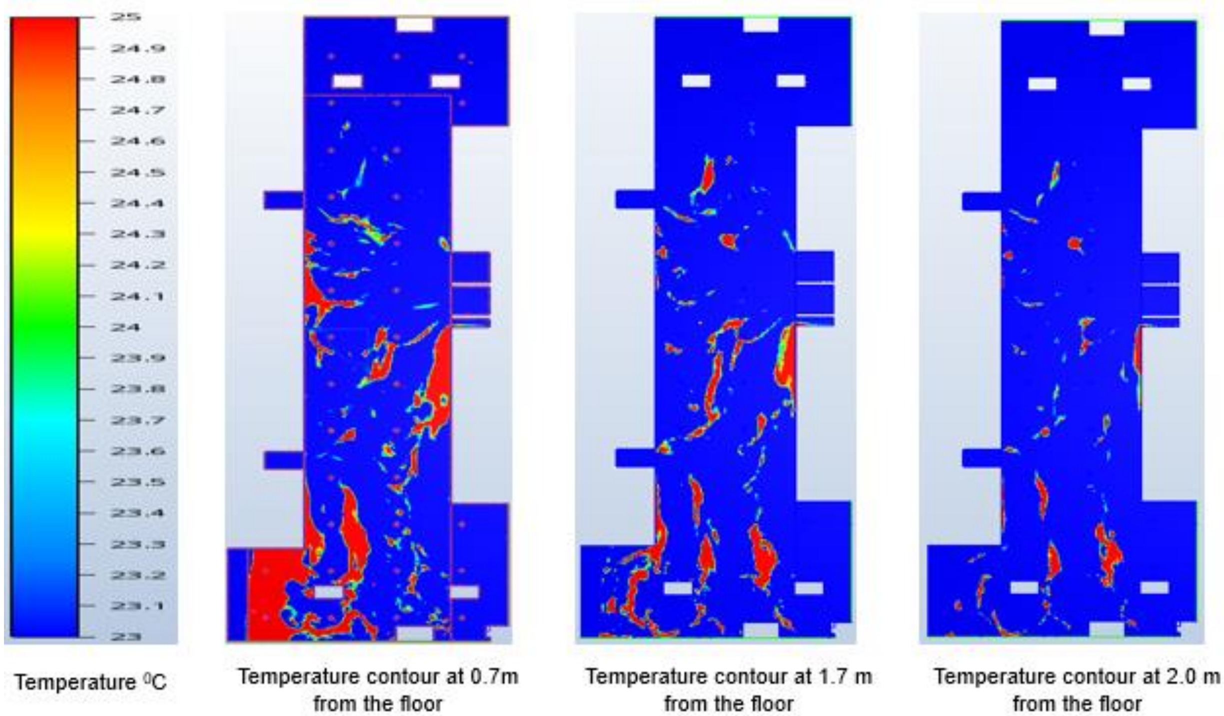



FIGURE 33: TEMPERATURE DISTRIBUTION (RESULTS FROM AUTODESK CFD USING PLANES OPTION)

*CONCLUSIONS*

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 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. Thus, Autodesk CFD Simulation and CFD results are used for:

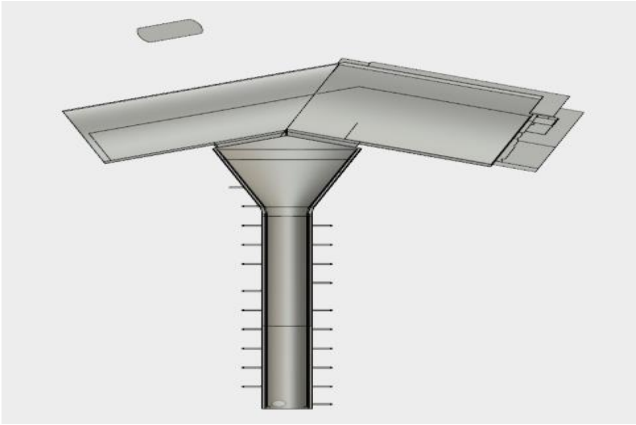
- Performance-based design of air-conditioning system
- addressing challenges imposed by architectural/ structural requirements
- evaluating thermal comfort of occupied space (e.g. Foyer) using Displacement Ventilation (DV)
- means of meeting sustainability goals by using energy efficient cooling system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

## Modeling and simulation of wind flow and rain flow on airport building

Wind flow over and around buildings create wind pressure on building elements such as roof and facades and hence exert wind load on the building structure. Also wind driven rain into buildings can be unwelcome and cause inconvenience to people and damage to building interior. Due to complex geometry and large size of modern buildings, it is not always possible to use analytical approach or plan for physical testing to determine effects of wind loading or rain ingress. In the following project examples, details of how Autodesk CFD simulation and CFD results can be useful in evaluating (i) wind pressures and (ii) rain ingress are described.

### (i) Wind loading analysis to simulate wind pressure and structural loading on airport façade and roof

Autodesk CFD can be used to evaluate wind pressure on airport façade and roof to analyze resultant structural loading. Modeling strategies include preparing a simulation ready CAD model either from Revit or Simstudio. While creating a CAD model for CFD, it is important to simplify the geometry which includes removal of features/details not relevant to analysis goals or features too small so that CFD simulation can be efficient in terms of time taken and computer hardware requirements.



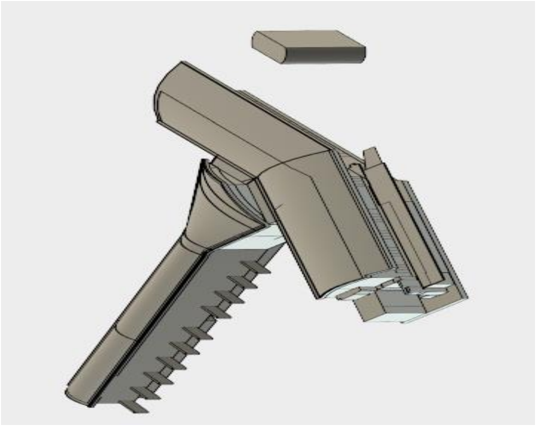


FIGURE 34: SIMULATION READY CAD MODEL OF AIRPORT TERMINAL BUILDING

Once a simulation ready CAD model of the building is prepared, external air volume surrounding the building has to be added to the model. External air volume represents open ambient environment. External air volume can be added either during CAD model preparation using the CAD tool or can be built in Autodesk CFD with the **External Volume** tool. In this case, external volume is added as a CAD part while preparing the CAD model of the building as it ensures the volume is coplanar with the ground-plane of the building.

External air volume geometry is built using Autodesk CFD recommended dimensions:

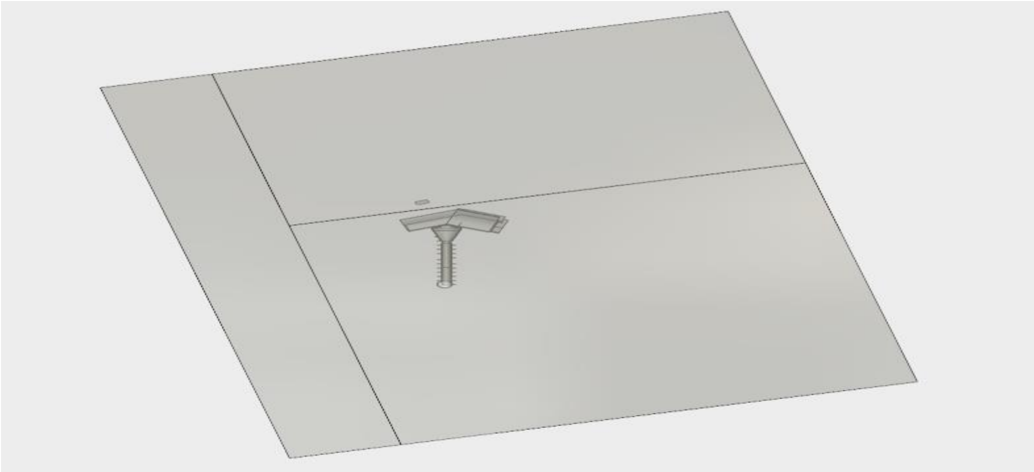


FIGURE 35: EXTERNAL AIR VOLUME CREATED IN CAD MODEL TO CAPTURE WIND FLOW OVER AND AROUND THE BUILDING

When the CAD model is transferred to Autodesk CFD, CFD model is ready for setting up simulation workflow. Simulation workflow includes assigning *materials, boundary conditions for wind flow, meshing, solver settings and results extraction*. External volume is assigned "*Air*" material with constant density as **buoyancy effects** are negligible. Building elements assigned as *solid* material are suppressed as we are interested in external wind flow over the airport building façade and roof.

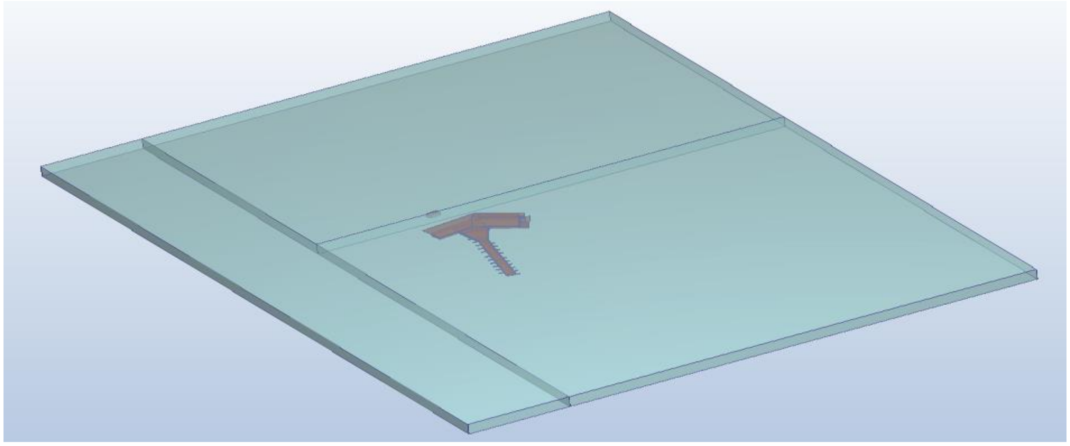


FIGURE 36: CFD MODEL WITH EXTERNAL VOLUME ASSIGNED "AIR" MATERIAL

Boundary conditions for wind flow analysis include assigning *wind speed* as a *velocity* boundary condition at the inlet of the air volume, as shown in Figure 37. Outlet of the air volume is defined by assigning *static gage pressure = 0*, as shown in Figure 38. Since the external volume is a free space, *slip/symmetry* boundary condition is applied to the top and sides of the air volume, as shown in Figure 39. Ground plane does not need to be specified with any condition as the air does not move along the ground plane.

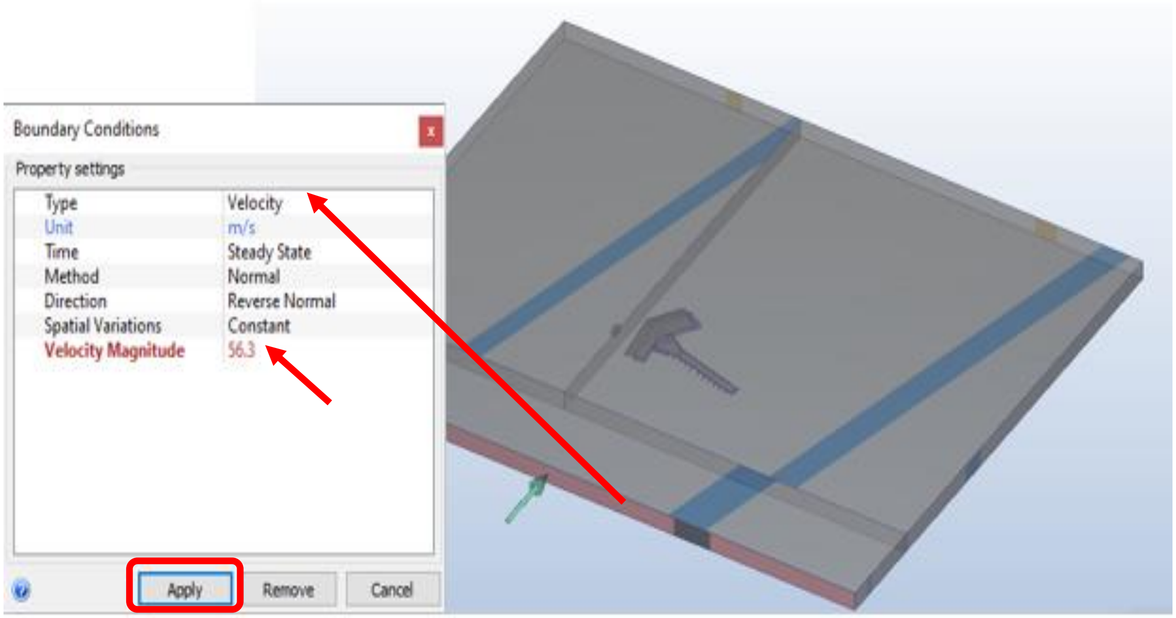



FIGURE 37: INLET BC (WIND VELOCITY) FOR EXTERNAL AIR VOLUME

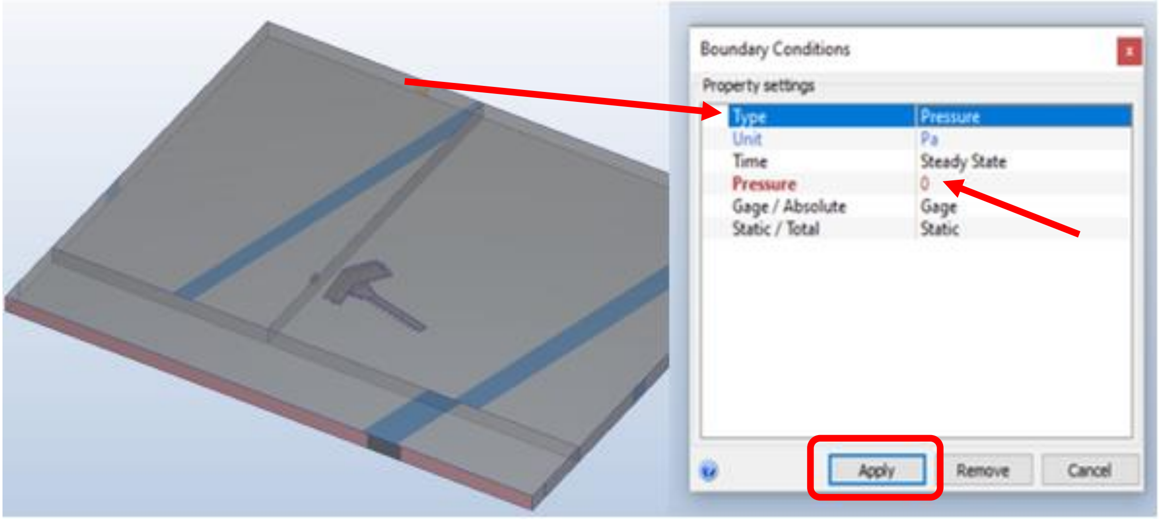



FIGURE 38: OUTLET BC FOR EXTERNAL AIR VOLUME

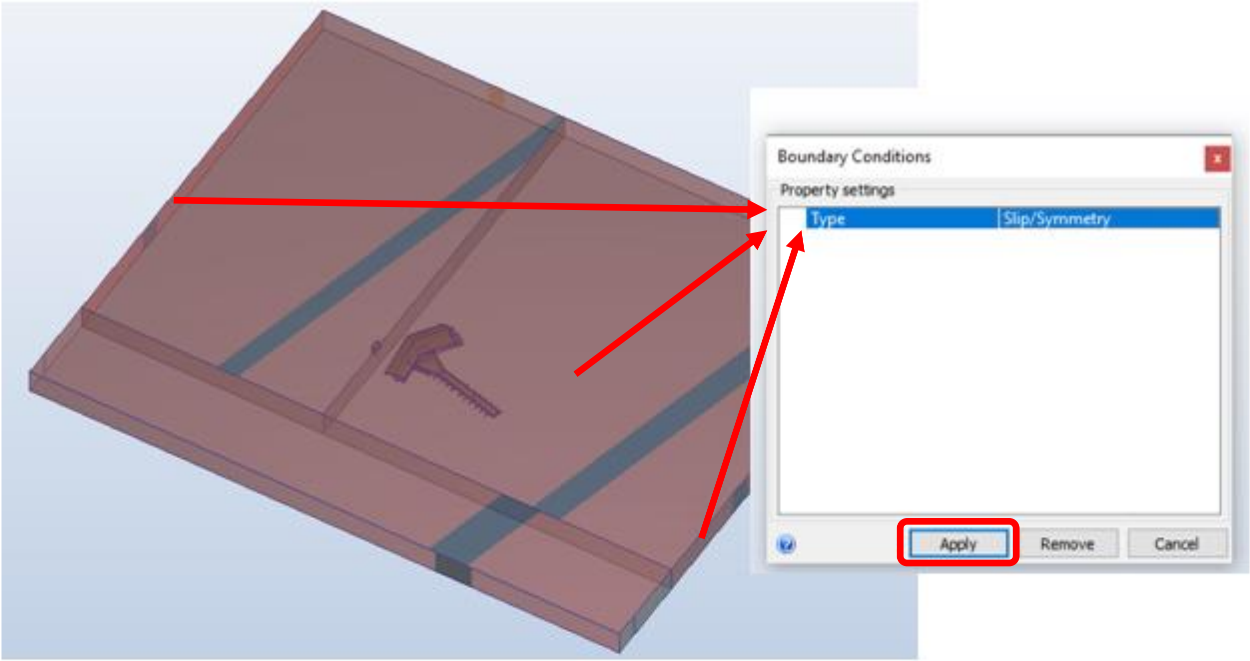



FIGURE 39: SLIP/SYMMETRY BC FOR EXTERNAL AIR VOLUMES FREE SPACES

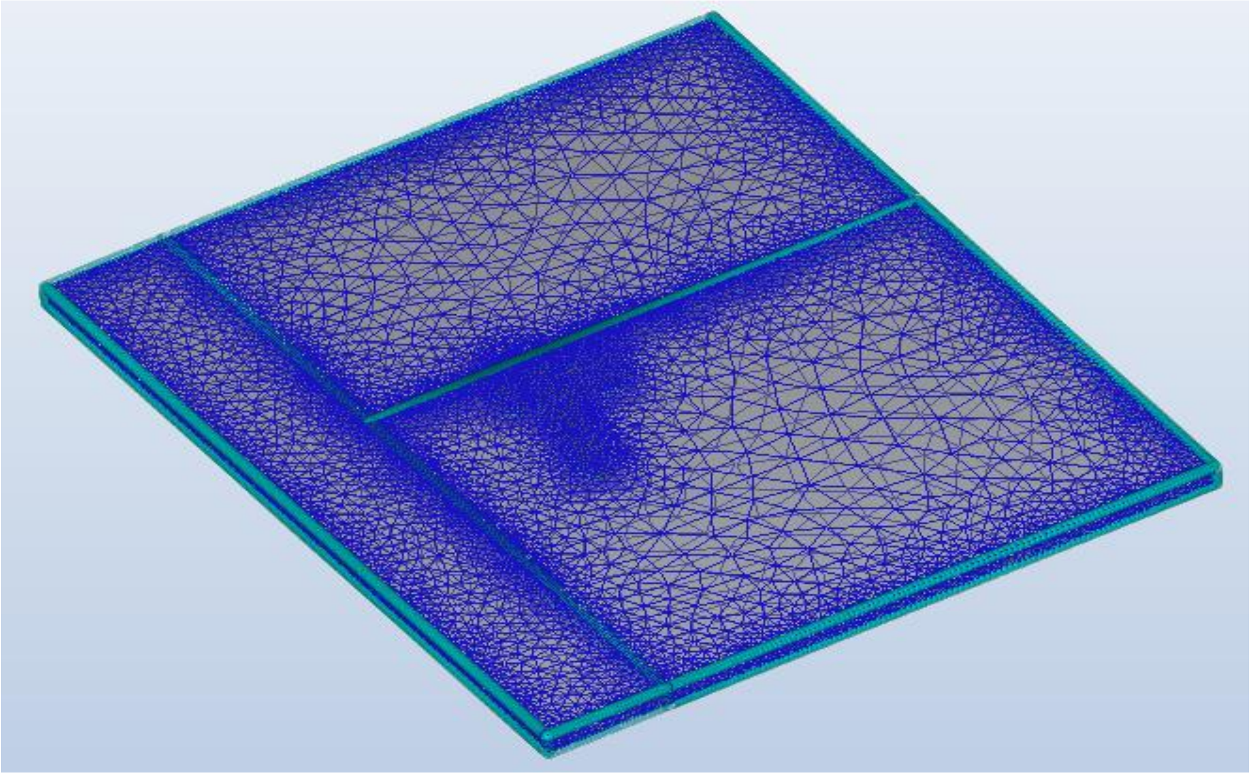


FIGURE 40: AUTOMATIC MESH SIZING WITH REFINEMENT

*Automatic mesh sizing* is used to define the mesh distribution. *Mesh refinement* is used on the geometric volume to resolve the flow gradients especially near the building in the upstream and downstream directions to capture flow gradients and flow separations, (see Figure 40).

With "*Flow = ON"* in the *Solve* dialog of Autodesk CFD, simulation is run till convergence and results extracted. *Particle traces* are a good way to visualize air movement around the building, as shown Figure 41 and Figure 42.

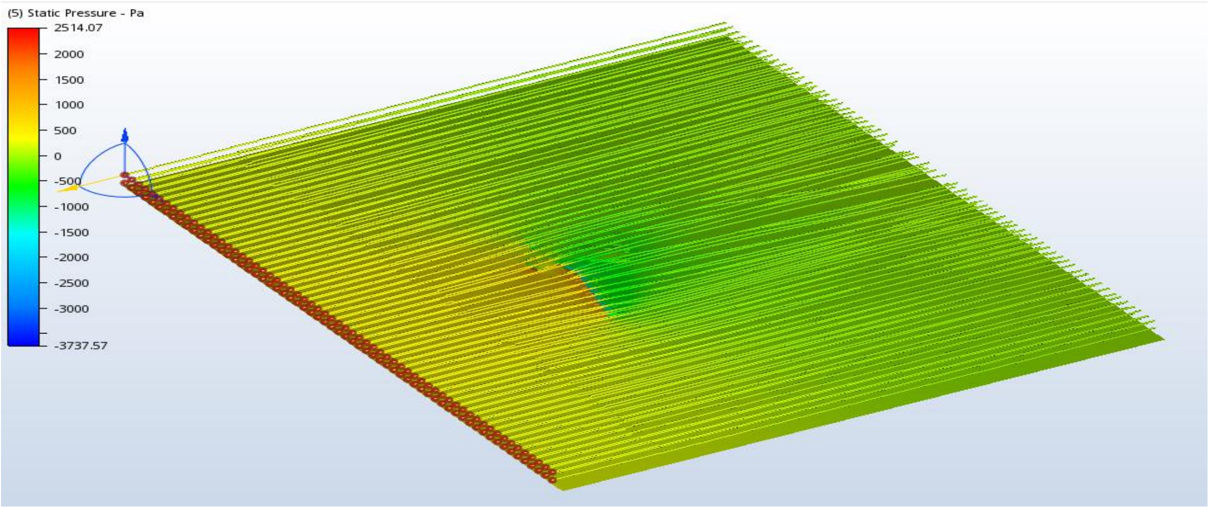



FIGURE 41: PARTICLE TRACES TO VISUALIZE AIR MOVEMENT OVER THE BUILDING

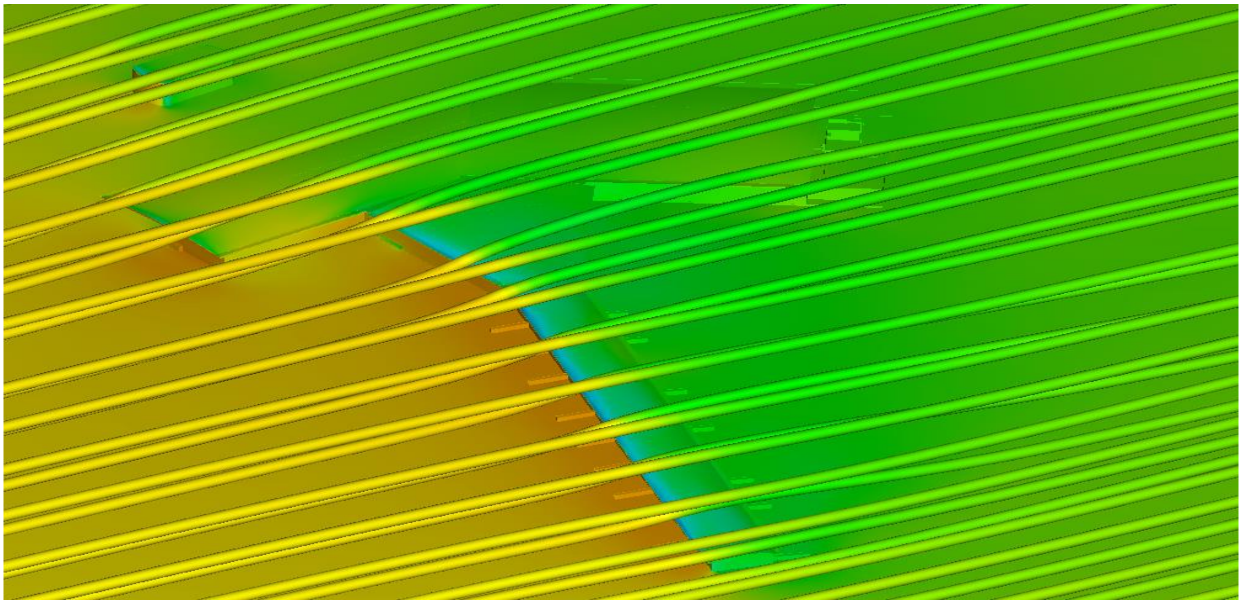


FIGURE 42: PARTICLE TRACES TO VISUALIZE AIR MOVEMENT OVER THE BUILDING

*Global Results* for static pressure are a good way to visualize wind pressure distribution on the airport building façade and roof, as shown in Figure 43 and Figure 44.

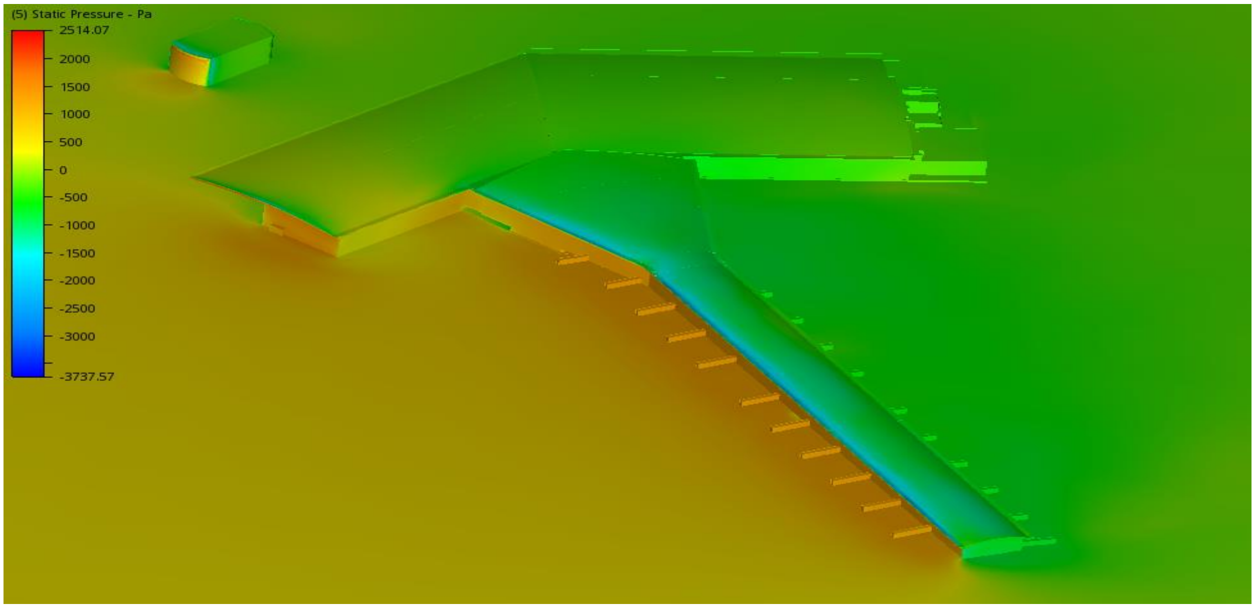



FIGURE 43: GLOBAL RESULTS FOR WIND STATIC PRESSURE

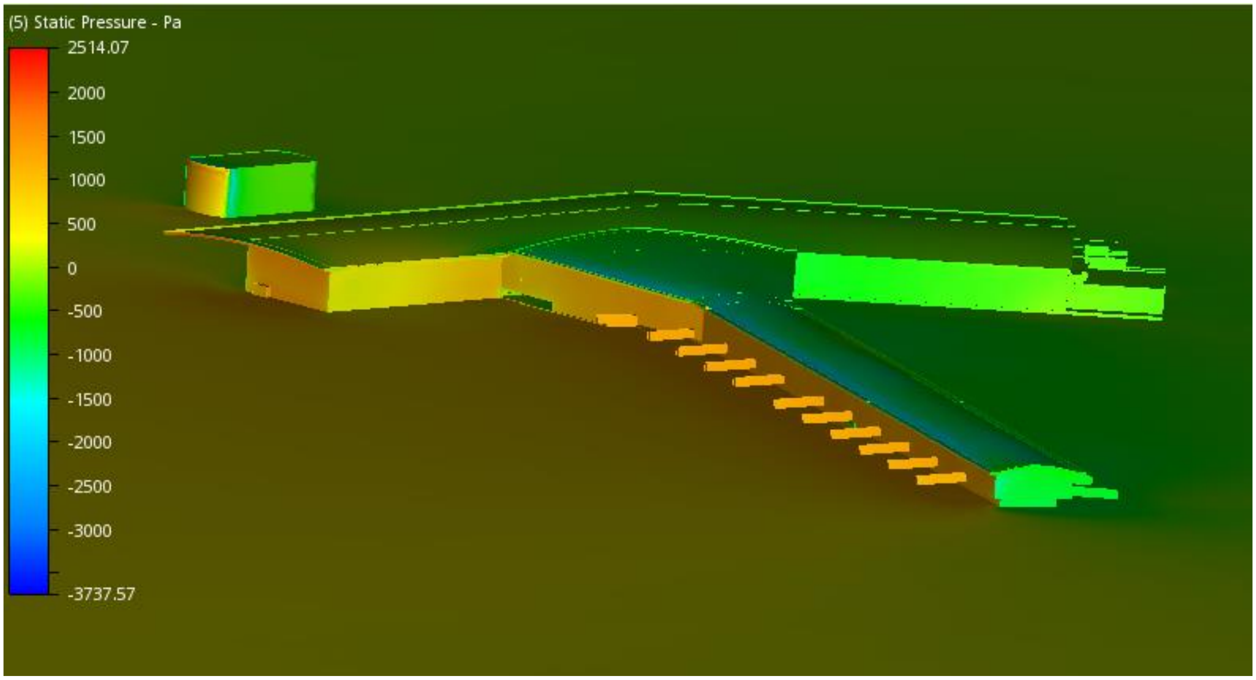



FIGURE 44: GLOBAL RESULTS FOR WIND STATIC PRESSURE

Using *Results Planes*, static pressure values are extracted at locations of interest in the building which represent wind loading effect, as shown in Figure 45 and Figure 46.

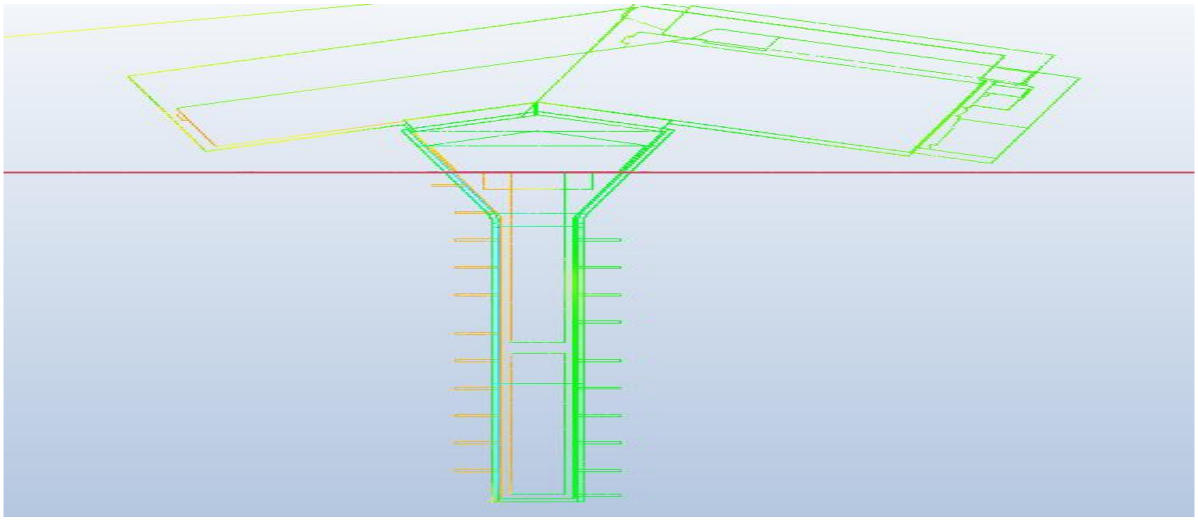


FIGURE 45: USING RESULTS PLANES OPTION FOR VISUALIZING WIND STATIC PRESSURE

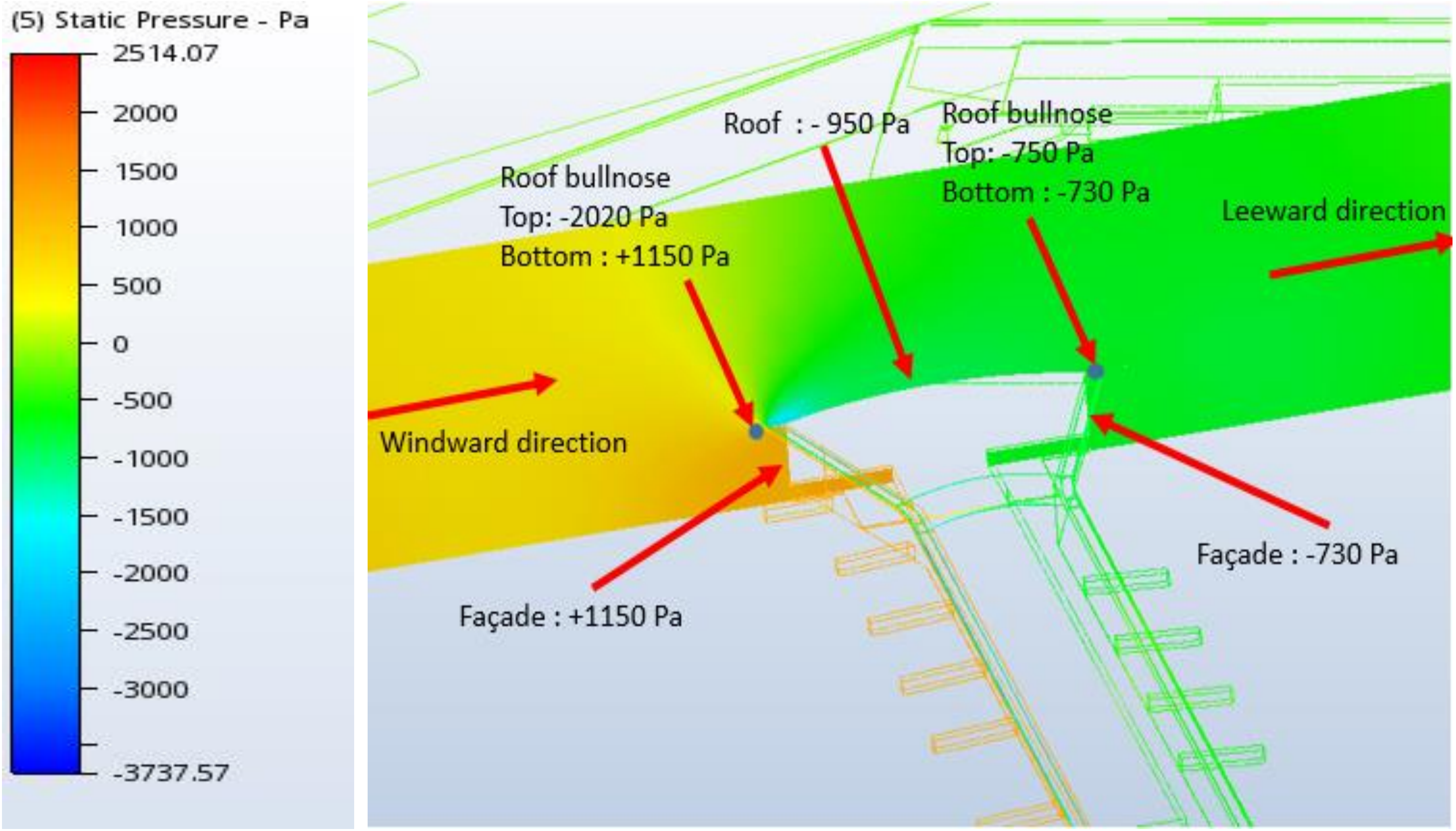



FIGURE 46: STATIC PRESSURE VALUES USING RESULTS PLANES

***CONCLUSIONS***

Autodesk CFD simulation and CFD results provide useful information on wind flow, resultant wind pressure on the building façade and roof, thereby a means of evaluating structural loading. Based on CFD results, designers can check if design is adequate or needs modification. Also CFD results provide opportunities to optimize the design towards material savings and meeting sustainability requirements.

## (ii) Rainwater ingress into airport building

Wind driven *rainwater ingress* into airport building is a challenging phenomenon, difficult to evaluate or determine its effects using traditional empirical approach due to complex airport roof geometry. However, using Autodesk CFD simulation, it is possible to determine airflow velocities and possibility of accompanying rainwater ingress.

Modeling strategies include preparing a simulation ready CAD model using Autodesk Fusion 360. Geometry is simplified to include only those geometry features affecting the airflow and rain ingress, as shown in Figure 47.

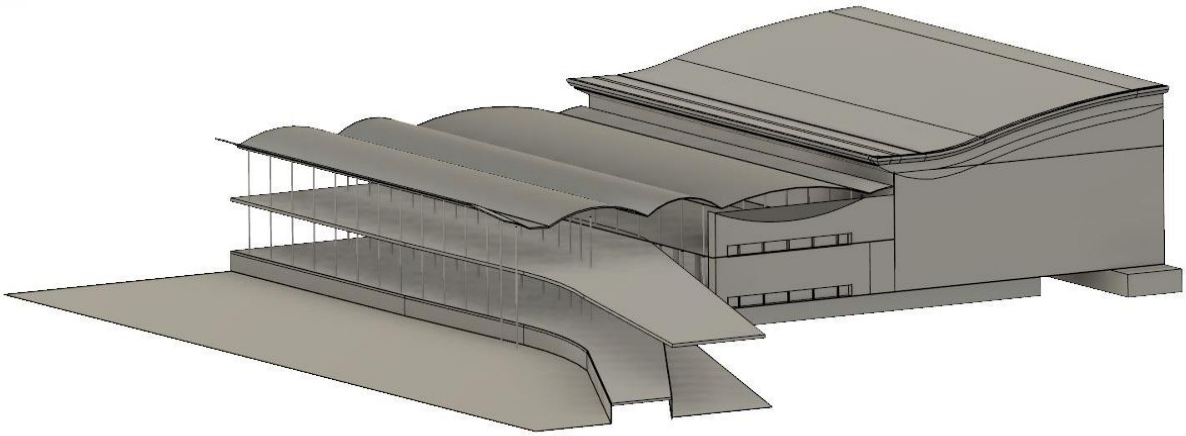


FIGURE 47: SIMULATION READY CAD MODEL OF AIRPORT TERMINAL BUILDING WITH FORECOURT

To simulate wind flow into and over the building, external volume geometry is prepared using Autodesk CFD **External Volume** tool.

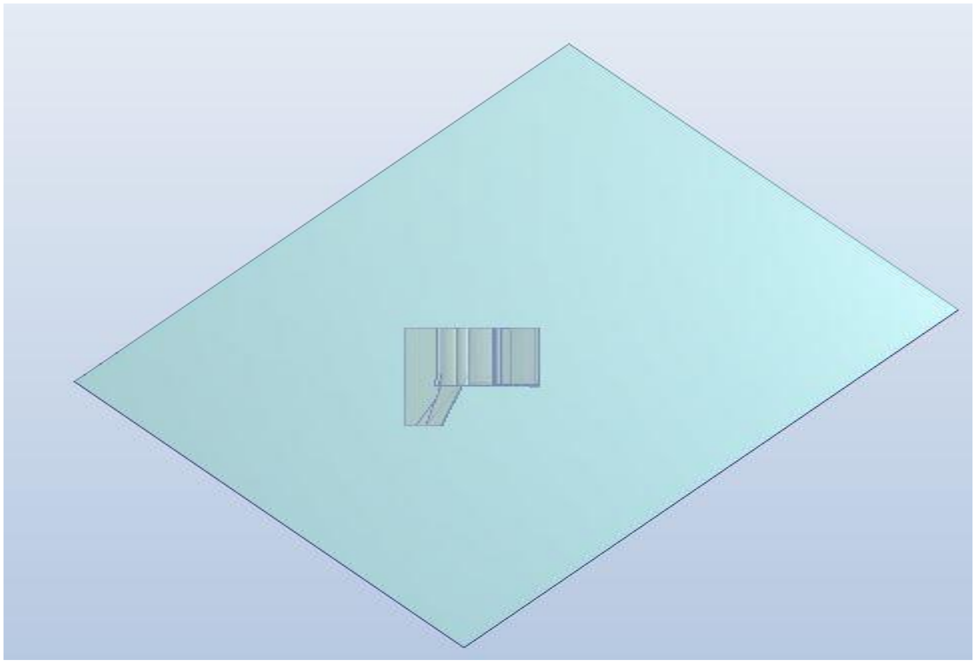


FIGURE 48: EXTERNAL AIR VOLUME CREATED IN AUTODESK CFD TO CAPTURE WIND FLOW INTO AND AROUND THE BUILDING

External volume is assigned "*Air*" material using **Materials** tab from the ribbon of **Setup** tab, (see Figure 49).

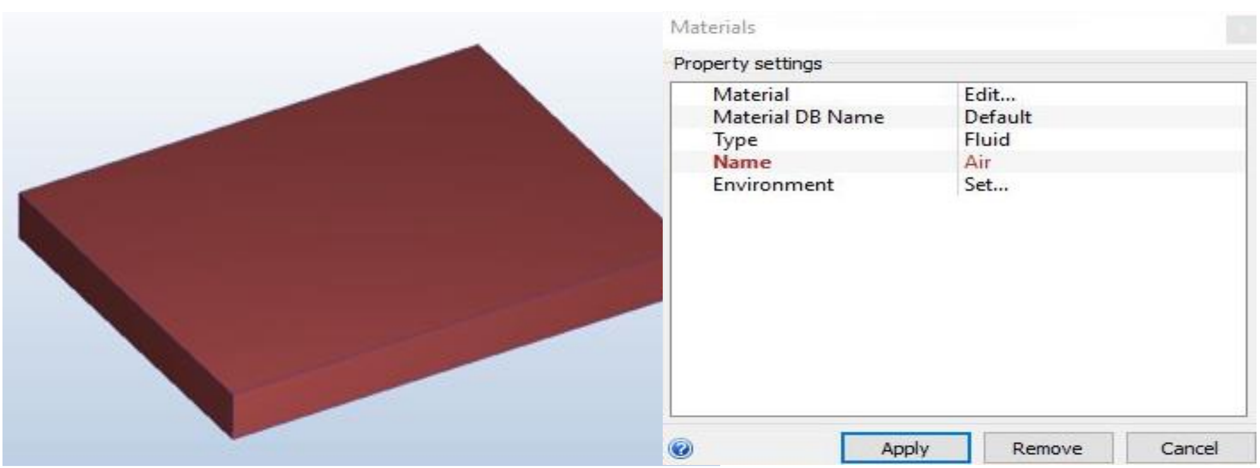




FIGURE 49: "AIR" MATERIAL ASSIGNED TO EXTERNAL AIR VOLUME

Building elements are assigned “*Concrete*” material and suppressed in the analysis, (see Figure 50)

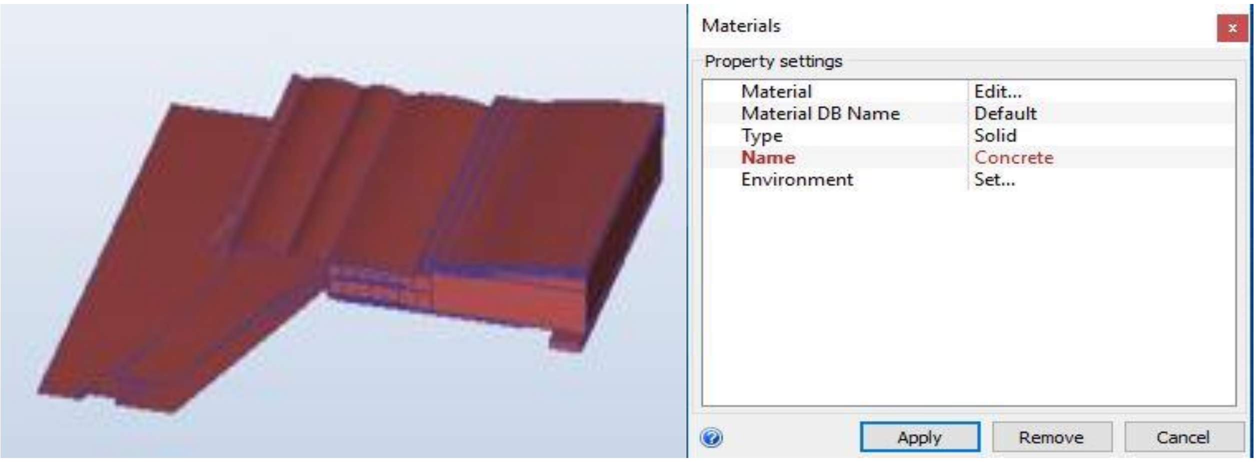



FIGURE 50: “CONCRETE” MATERIAL ASSIGNED TO BUILDING ELEMENTS

*Boundary conditions* for wind flow analysis include assigning *wind speed* as a *velocity* boundary condition at the inlet of the air volume, *static gage pressure* at the outlet of air volume and *slip/symmetry* for top and sides of the air volume, as shown in Figure 51.

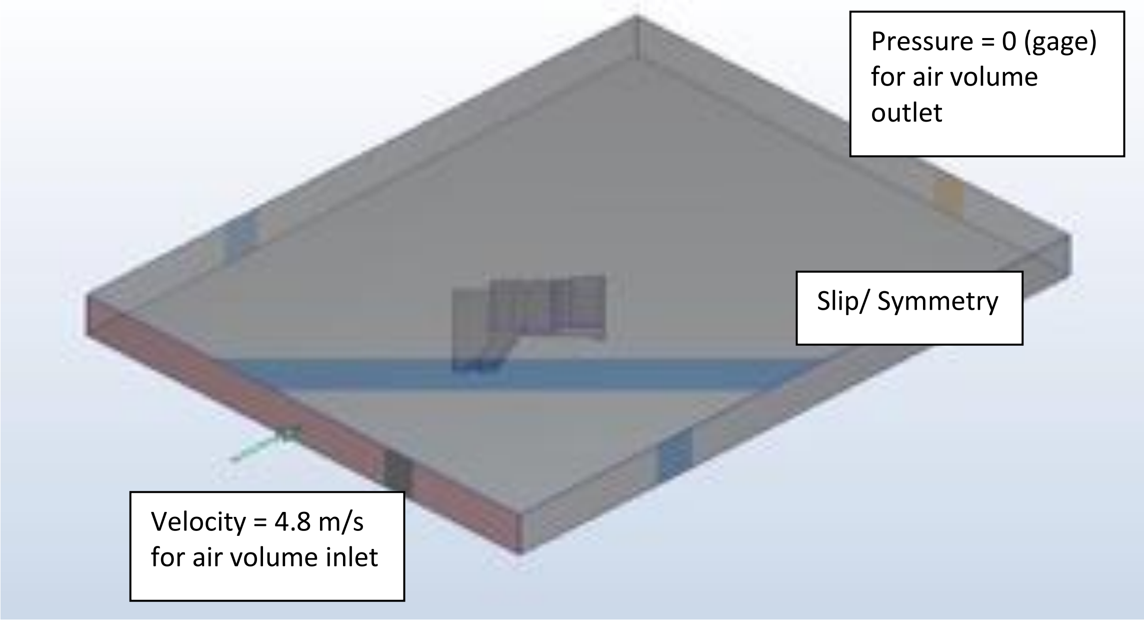



FIGURE 51: BOUNDARY CONDITIONS FOR CFD SIMULATION OF WIND FLOW

*Automatic mesh sizing* is used to define the mesh distribution, as shown in Figure 52. *Mesh refinement* is used on the geometric volume to resolve the flow gradients especially near the building in the upstream and downstream directions to capture flow gradients and flow separations.

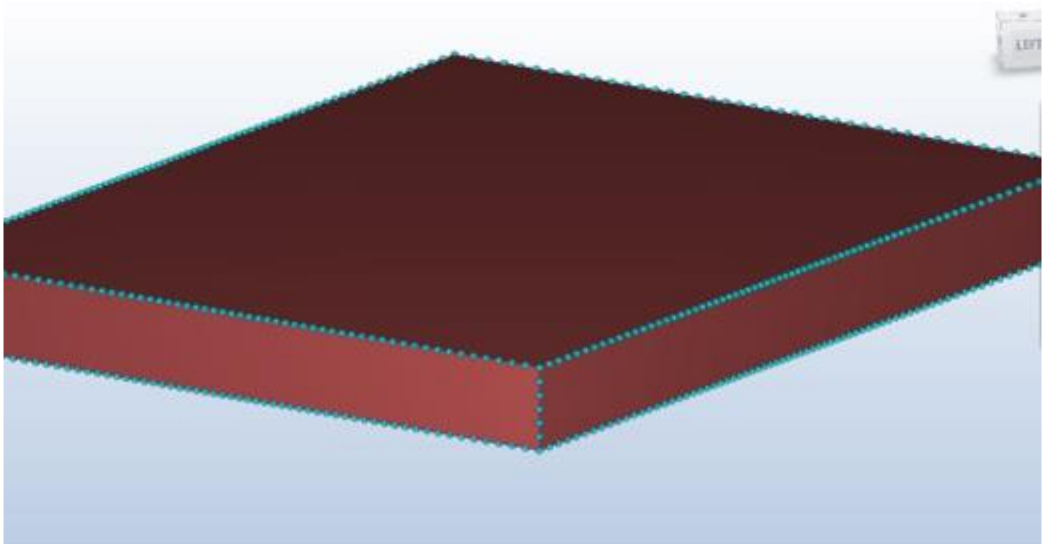


FIGURE 52: AUTOMATIC MESH SIZING WITH REFINEMENT

With "*Flow = ON"* in the *Solve* dialog of Autodesk CFD, simulation is run till convergence and results extracted. *Particle traces* are a good way to visualize air movement around the building, as shown Figure 53.

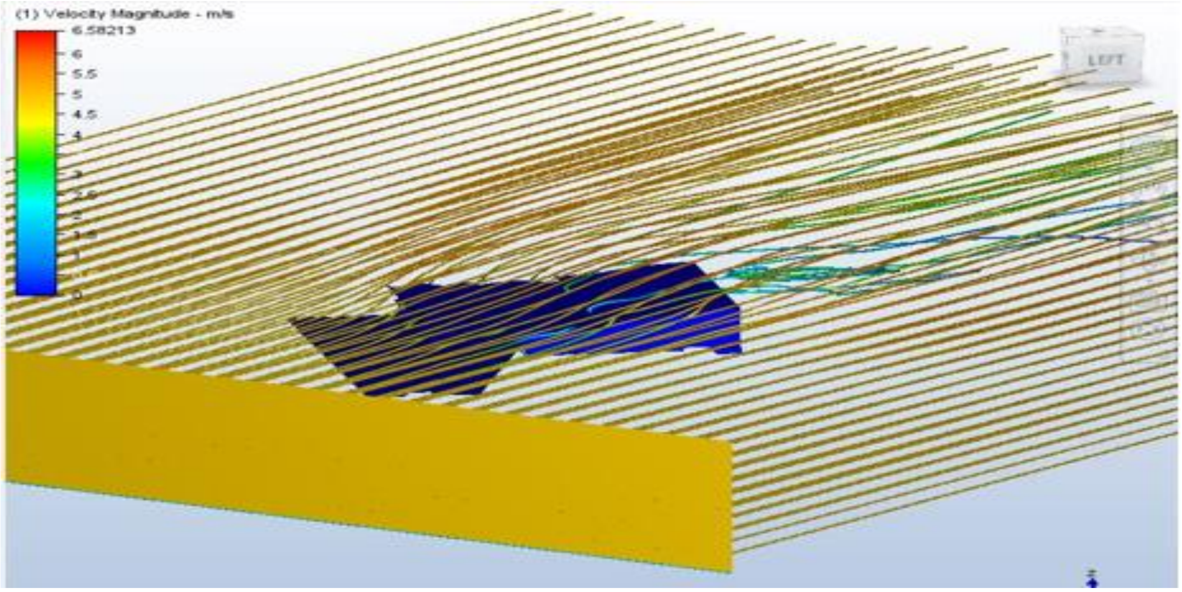



FIGURE 53: PARTICLE TRACES TO VISUALIZE AIR MOVEMENT OVER THE BUILDING

Using *Results Planes*, air flow velocity is extracted at location of interest in the building, as shown in Figure 54 to determine rain ingress into forecourt area of airport terminal building.

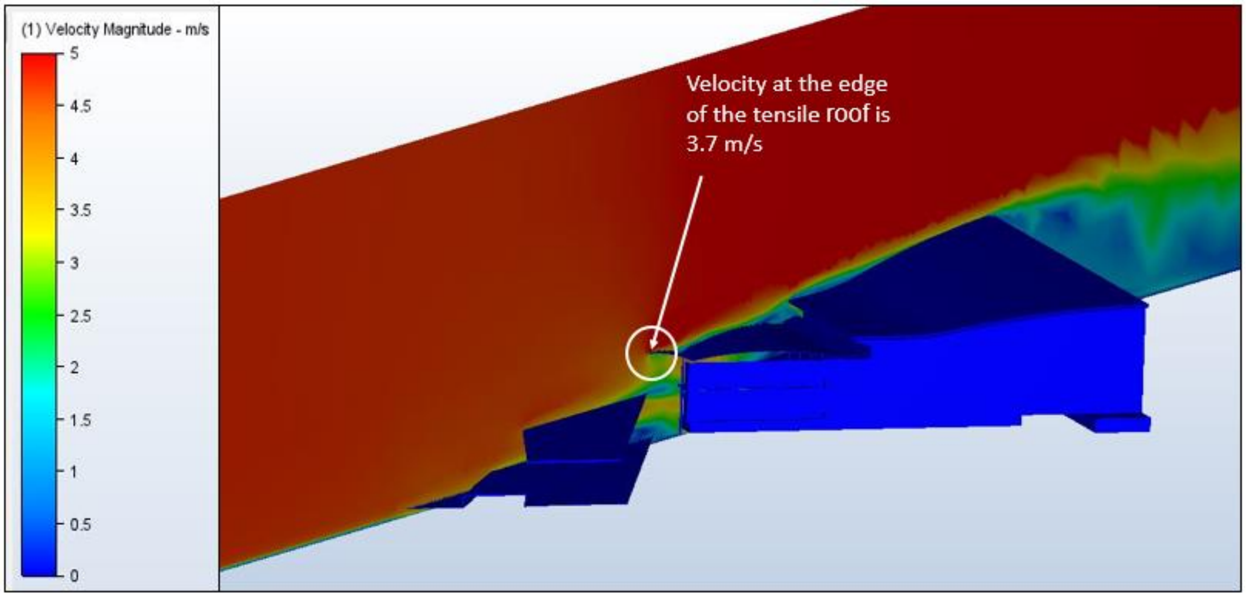



FIGURE 54: AIR FLOW VELOCITY NEAR ROOF EDGE OF FORECOURT AREA OF AIRPORT BUILDING

The rainwater ingress calculations based on air flow velocity obtained from Autodesk CFD results are given below and shown in Figure 55.

The relationship between intensity of rainfall and mean speed of falling rain drops is, $V = 4.5\ I^{0.107}$

Where V = Mean speed of rain drops (m/s).
I = Intensity of rainfall (mm/hr)

Then mean angle of deflection (ϴ) of the rain from vertical is given by: $\tan \Theta = U/V$
Where U is the speed of wind accompanying rainfall

$U = 3.7$ m/s, as per CFD simulation.
$I = 163$ mm/hr as per client data.
$V = 4.5\ (163)^{0.107} = 7.76$ m/s
$\tan \Theta = 3.7/7.76 = 0.4768$
$\Theta = 25.49$ deg.

These calculations indicate that the wind driven rain is not likely to wet the forecourt area.

*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*

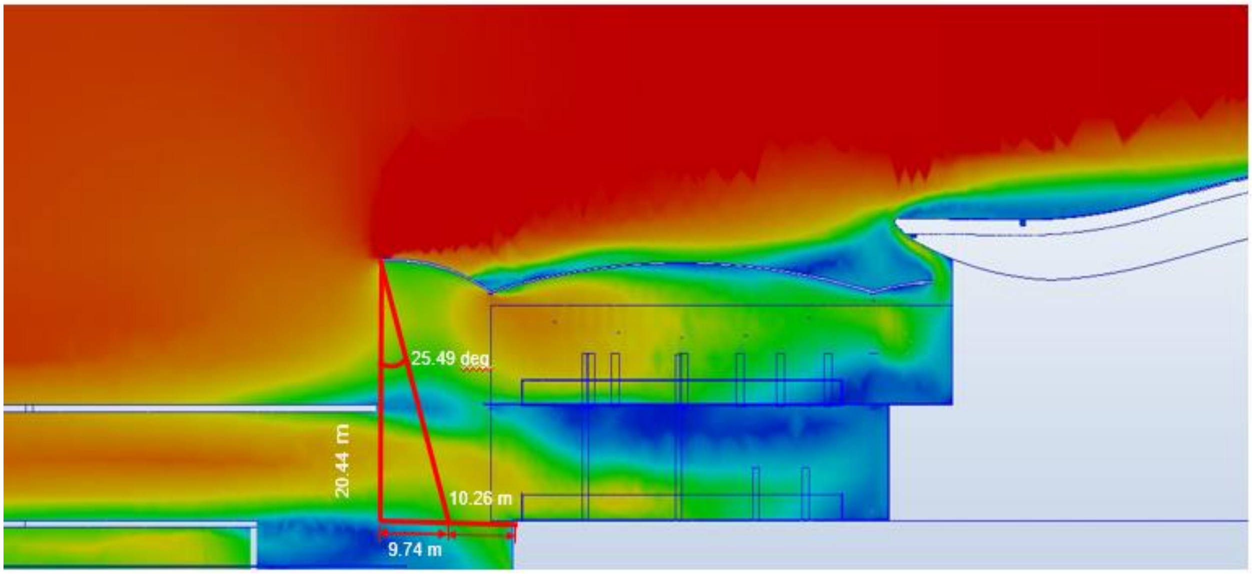



FIGURE 55: WIND DRIVEN RAINWATER DEFLECTION INTO FORECOURT AREA OF AIRPORT BUILDING

***CONCLUSIONS***

Autodesk CFD simulation and CFD results provide useful information on wind flow, resultant air velocity values on the airport roof and provides a useful method of estimating wind driven rainwater ingress into the building area of interest.

## Optimization of storm water roof drainage using free surface flow

Original bell mouth design for roof drainage (as per Tender specification, with circular bell mouth of 700mm diameter to 346mm diameter drain pipe) 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.

Autodesk CFD simulation was used 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.

Modified design was proposed with 750mm x 400mm bell mouth transitioning to 300mm dia. drain pipe. Modified bell mouth design was validated by CFD analysis for its adequacy in rainwater drainage for a 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.

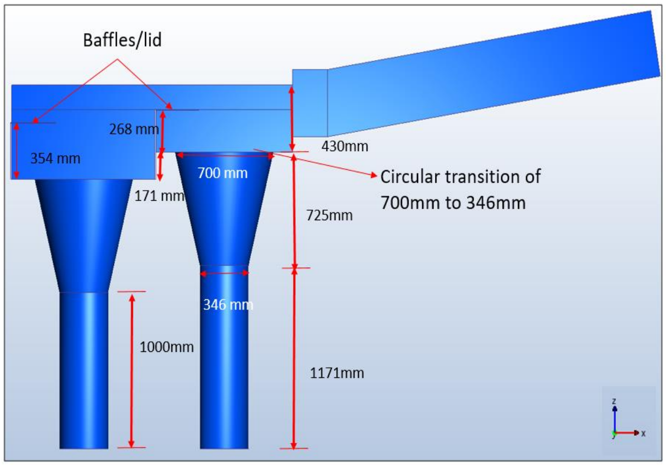



FIGURE 56: ORIGINAL DESIGN – RAINWATER DRAINAGE SYSTEM –AIRPORT ROOF

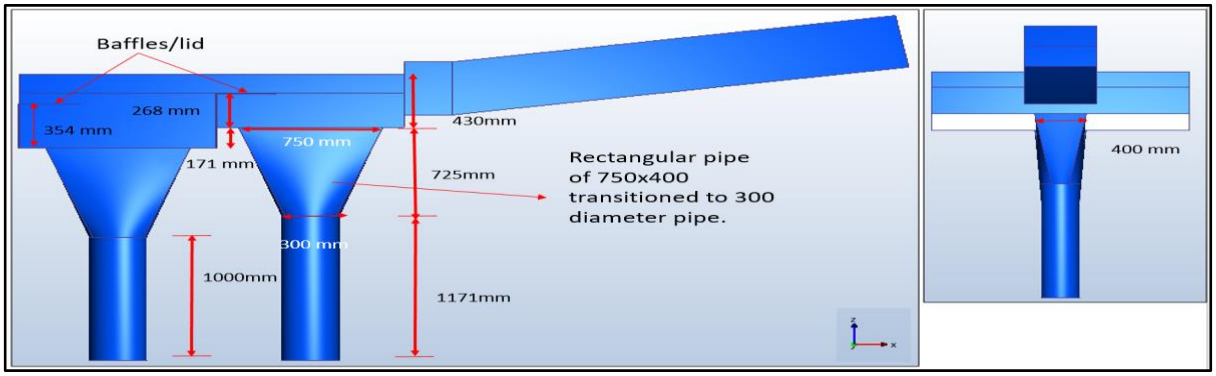


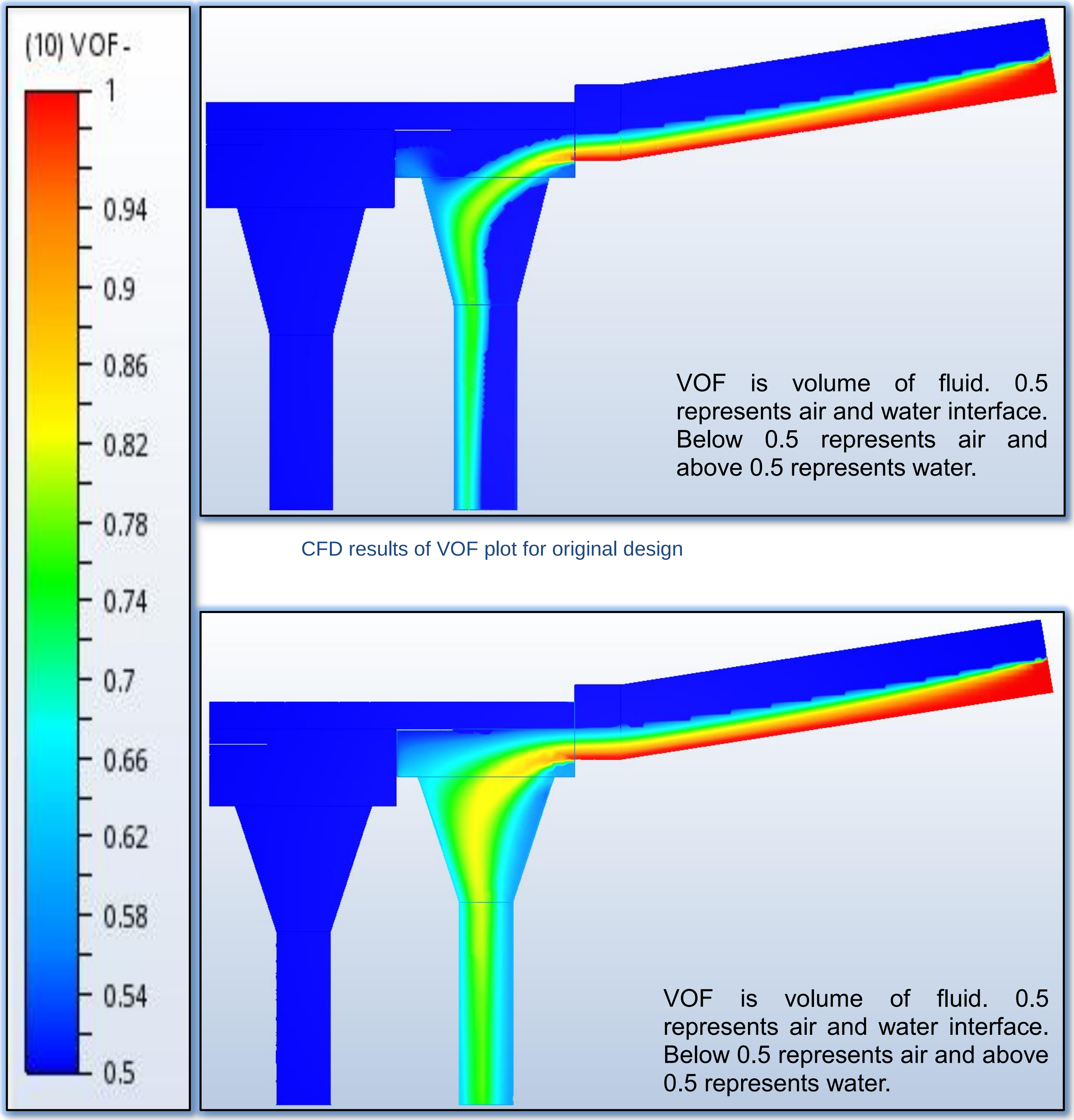

FIGURE 57: MODIFIED DESIGN – RAINWATER DRAINAGE SYSTEM –AIRPORT ROOF

3D model is prepared in Simstudio and imported into Autodesk CFD software. Collection box, bell mouth and drain pipes are modelled as per proposed design, as shown in Figure 57. **Free surface flow** analysis using **VOF** (Volume of Fluid) method is carried out using CFD software. Modified design scenario is analyzed to evaluate water flow and VOF distribution to arrive at optimum design for bell mouth and drain pipes for draining the rainwater.

*Original design*: As per Tender specification, with circular bell mouth pipe of 700mm diameter to 346mm diameter drain pipe, (see Figure 56).

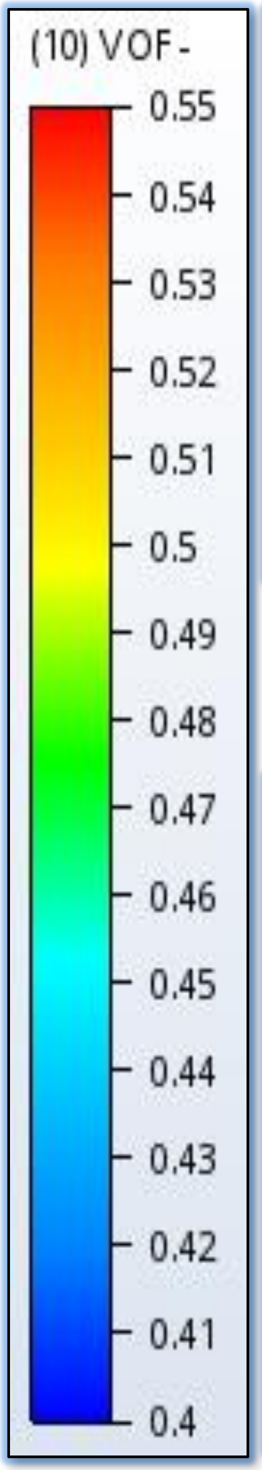
*Modified design*: *Rectangular bell mouth* of 750mm x400mm transitioned to 300mm drain pipe, (see Figure 57).

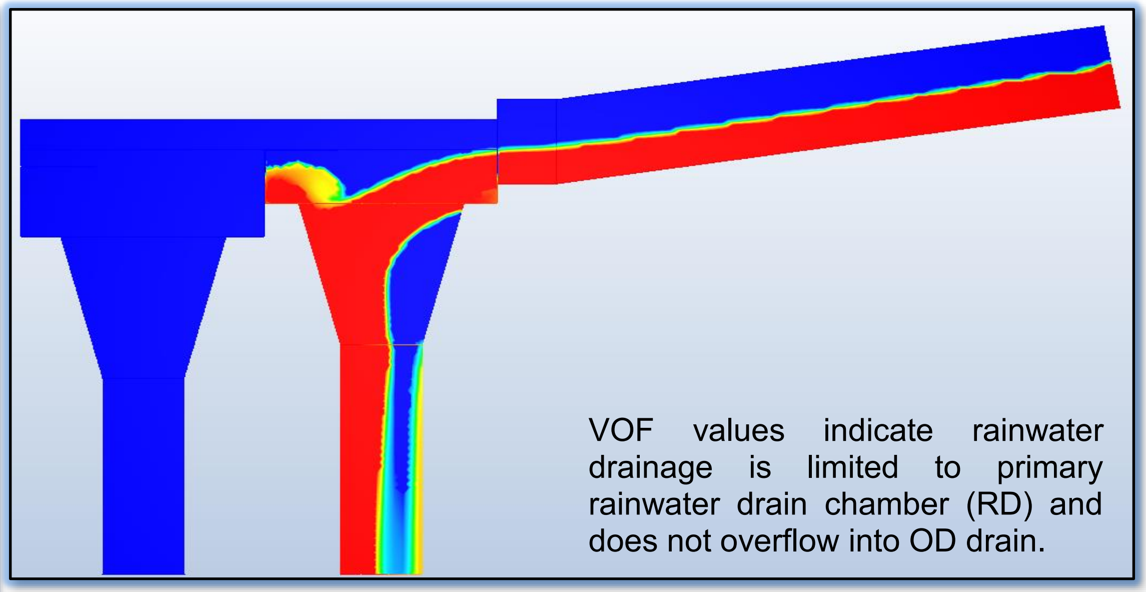
*Acceptance criterion*: For maximum rain intensity for site, *rainwater should not overflow the sump pit*.




CFD results of VOF plot for original design

FIGURE 58: CFD RESULTS VOF PLOT FOR MODIFIED DESIGN




CFD results of VOF plot for original design

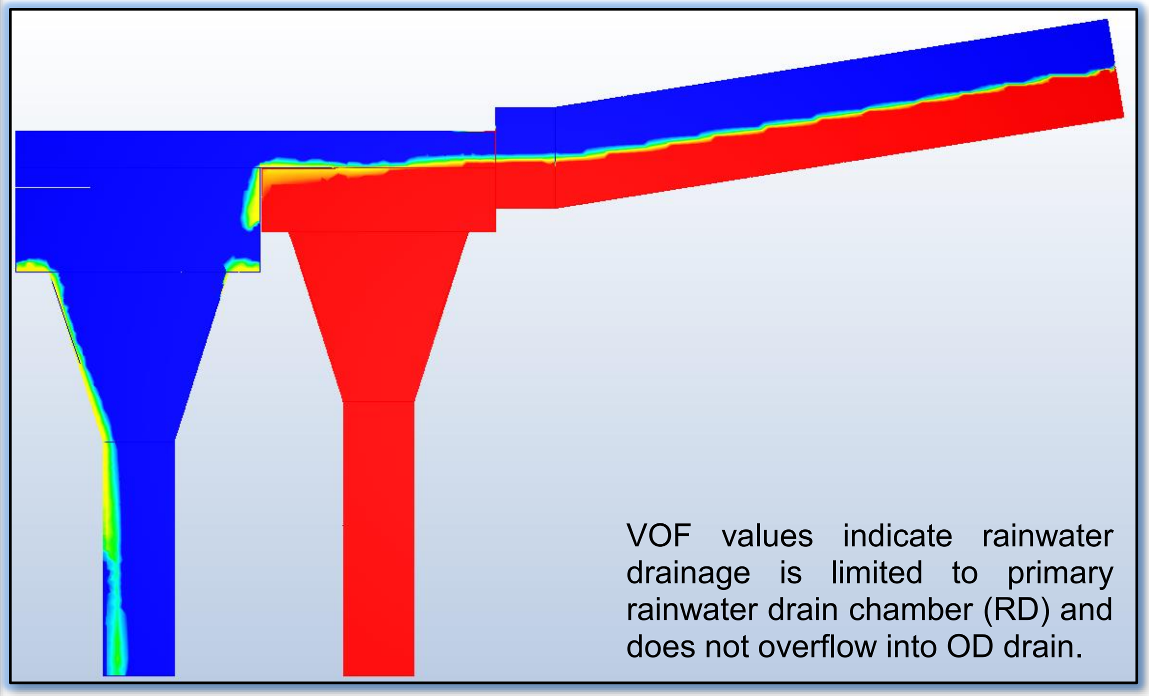



FIGURE 59: CFD RESULTS VOF PLOT FOR MODIFIED DESIGN

**RESULTS**

Figure 58 shows the **VOF** plots (range of 0.5 – 1) for both the original and modified design.
Figure 59 shows the **VOF** plots (range of 0.4 - 0.55) for both the original and modified design.

***CONCLUSIONS***

1. *Original design*: CFD results show that rainwater in the collection box is not flowing into the overflow drain chamber and entire rainwater is flowing through the primary rainwater drain (RD) chamber.
2. *Modified design*: CFD results show that the rainwater 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).
3. Since there is constructability issue of 700mm diameter bell mouth pipe, modified design was proposed with 750mm x 400mm bell mouth transitioned to 300mm dia. drain pipe which gives acceptable results same as original design.
4. Based on the CFD analysis, modified bell mouth size is recommended to accommodate constructability issue

## Relevant results from CFD for sustainable design

- *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

# Thank you attending and have a great AU 2020!

*Acknowledgements: Greatful to Mr. K. Veerappan, VP & HEAD- EDRC and Mr. R. Balakrishnan, GM & HEAD- MEP Design, L&T Construction, Larsen & Toubro Limited for giving me opportunity to participate and present at AU2020 and Srikanth Modem and Anand Doss for their help in CFD simulation.*