BLD223707

Make Your MEP Design Better using Autodesk CFD Simulation

Dr. Munirajulu M
Chief Engineering Manager, MEP-CFD
L&T Construction, Larsen & Toubro Limited, Chennai, India

Learning Objectives

- Learn how air flow and heat transfer can be modeled in MEP design analysis
- Understand application-specific simulation strategies and techniques
- Learn how to highlight key results from Autodesk CFD analysis to demonstrate MEP design performance
- Discover advantages and limitations of using Autodesk CFD for MEP design

Description

Autodesk CFD software is a great simulation tool for solving practical problems of MEP (mechanical, electrical, and plumbing) design for commercial buildings. In this class, you will see simulation strategies and techniques as applied to MEP design analysis to achieve better designs. We will share our experience of actual projects where Autodesk CFD has been used to zero in on effective ventilation in a diesel generator room; to gain insight to arrive at desirable smoke exhaust design for life safety in a public space; to improve design layout of a basement car park for acceptable ventilation; and to troubleshoot water leakage in an airport roof by rainwater flow simulation. You will learn how airflow and thermal simulations from Autodesk CFD are used to demonstrate the performance of MEP design. You will understand how results visualization and extraction tools are useful for spotting design issues and fixing them cost-effectively. We will also highlight advantages, best practices, and limitations while using Autodesk CFD for the MEP design process.

Speaker

Dr. Munirajulu. M, B.Tech and Ph.D. from IIT, Kharagpur, India, has more than 22 years of direct and indirect involvement with CFD technology as a design analysis tool in areas such as HVAC, Automotive, Fluid Handling Equipment, Steam turbines and boilers. 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. His professional interests include state of the art CAE technologies (CAD, CFD and FEA). Currently he is responsible for CFD analysis in MEP design related to commercial buildings and airports in L&T Construction, Larsen & Toubro Limited, Chennai. He has been using Autodesk CFD Simulation software for HVAC and MEP applications in areas such as thermal comfort, data center cooling, basement car park ventilation, DG room ventilation effectiveness, rain water free surface flow for roof design, and smoke simulation in buildings in design stage as well as for trouble shooting. He has published 4 nos. of technical papers in international journals of repute and has been a speaker at technical conferences including AU.

Contents

Air flow/ fluid flow and heat transfer modelling in MEP design analysis ... 4
Airflow / fluid flow modelling ... 4
Heat transfer modelling ... 4
Free surface flow modelling ... 5
Simulation strategies and techniques-DG room ventilation ... 6
CAD model ... 7
Materials assignment ... 8
Boundary Conditions (BCs) ... 10
Meshing ... 11
Solver settings ... 11
Key results for design performance –DG room ventilation ... 12
CFD results for Original Design (as-was lay out) ... 14
CFD results for Design Option 1 ... 15
CFD results for Design Option 2 ... 16
Simulation strategies and techniques- Smoke visibility and extraction ... 17
Smoke visibility ... 17
CAD model ... 17
Materials assignment ... 18
Boundary Conditions (BCs) ... 19
Meshing ... 20
Solver settings ... 20
Key results for design performance –Smoke visibility and extraction ... 22
Smoke visibility results ... 23
Smoke temperature results ... 24
Air / Smoke flow field results ... 25
Simulation strategies and techniques- Basement car park ventilation ... 26
CAD model ... 26
Materials assignment ... 27
Boundary Conditions (BCs) ... 27
Solver settings ... 27
Key results for design performance –Basement car park ventilation ... 27
LMA values for Initial Design ... 28
LMA values for Design Options 1-4 ... 29
Simulation strategies and techniques – Rain water flow simulation ... 30
CAD model ... 30

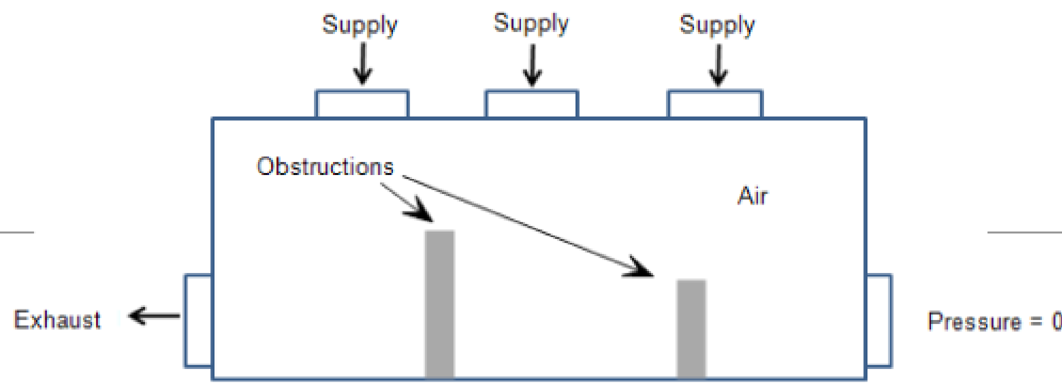
Free surface flow modelling 31
Key results for design performance –Rain water flow simulation........ 33
VOF results for rain water level- Initial Design and Final Design 33
Advantages and limitations of using Autodesk CFD for MEP design........ 34
Advantages........ 34
Limitations........ 34
References 34

Air flow/ fluid flow and heat transfer modelling in MEP design analysis

MEP design includes many systems where air flow and heat transfer take place and design goals focus on ensuring proper air/water flow and temperature control within the building space, hence plays a vital role in design and construction of commercial, residential and industrial buildings.

Airflow / fluid flow modelling

Air management systems include a network of supply and returns either for ventilation or occupant air conditioning. Air or water flow in MEP systems are incompressible (Mach No<0.3) and Autodesk CFD has option to select type of flow (incompressible/ compressible) during model set up. Typical boundary conditions include pressure inlet and exhaust flow outlet or flow inlet with pressure outlet.



Typical airflow modeling

Air flow is modeled with air region and internal obstructions. Supply includes vents, diffusers, fans etc. and exhaust includes registers, vents, fans etc.

https://knowledge.autodesk.com/search?search=mechanical%20ventilation&p=SCDSE&sort=score

Tips: Extending Inlet and Outlets to Help Avoid Divergence, Improve Mass Balance and Aid Accuracy

Heat transfer modelling

Heat transfer is modeled as: forced convection, mixed convection, natural convection and conjugate heat transfer.

Forced convection

In a forced convection analysis, the flow and heat transfer can be solved separately because the flow does not depend on the temperature distribution. An often used technique is to compute the flow solution prior to computing the thermal distribution. Unlike a buoyancy-driven solution, the flow and heat transfer solutions are decoupled from one another.

Mixed convection

In many MEP applications such as smoke analysis, DG room ventilation and basement ventilation, cooler air is blown but may contain local temperature gradients that will cause some appreciable buoyancy effects. This type of heat transfer is known as mixed convection, since it has features of both natural and forced convection.

Natural convection

Natural convection occurs as a result of buoyancy-driven flow caused by density gradients due to temperature variations. Examples include natural ventilation for open car park or smoke removal in a large space without using any fans or blowers. Buoyancy driven flow convects heat from heated source/components and removes it from built space. Both flow and heat transfer are solved simultaneously as thermal and flow physics are coupled.

Conjugate heat transfer

Solid material conduction and the fluid convection are analyzed simultaneously. The type of fluid convection (natural, forced or mixed) determines the analysis parameters.

Free surface flow modelling

Free Surface modeling capability dynamically simulates the interface between liquids and gases, capturing flow phenomena such as waves, sloshing, and spilling. One of the engineering applications in MEP is modeling and simulation of rain water flow on airport roofs with a view to avoid water leakage into occupied spaces through skylights.

The Free Surface formulation in Autodesk® CFD is based on the Eulerian Volume of Fluid (VOF) approach.
Free Surface simulations are always run as time dependent and use default settings in Autodesk CFD.

- On the free surface (fluid-gas interface), the static gage pressure = 0
- Gas-filled regions are simulated by empty elements.
- These empty elements are omitted from the flow solution thus providing interface between liquid and gas

In the following sections, simulation strategies and techniques as applied to DG room ventilation for power backup, smoke removal in case of fire situation, contaminants removal from basement car park and the rain water leakage into occupied space in the building, are addressed.

Simulation strategies and techniques-DG room ventilation

It was observed that DG sets were having an issue of tripping when loaded to more than 50% rated capacity. DG room temperature was noted to be rising sharply when DG sets were run. Less/inadequate air movement over DG for cooling alternator and engine was thought to be the reason for DG failure/ performance issues.

DG room ventilation design goal is to ensure required air flow distribution over heat radiating components like alternator, engine and radiator coil. Proper air flow results in acceptable room temperature for service personnel as well as component temperatures to avoid derating of DG equipment.

Goal of CFD simulation is to determine ventilation air flow and temperature distribution in the DG room taking into account heat generated from engine, alternator and radiator coil (heat rejection to atmosphere)

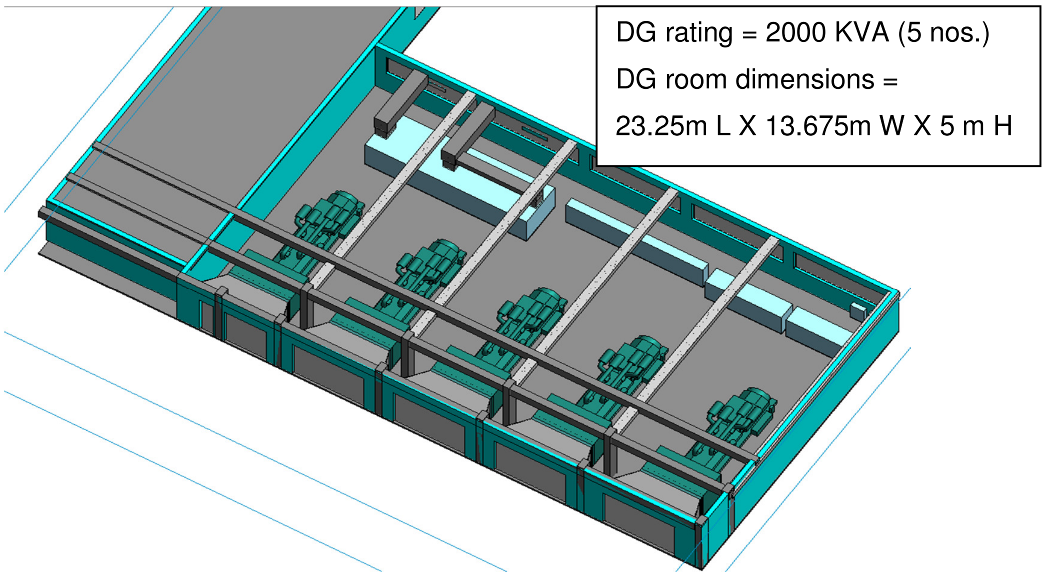
Design performance criteria:

- Air temperature around the engine will not exceed 50^0 C and 45^0 C around alternator.
- Bottom to top air flow from the ventilation system.

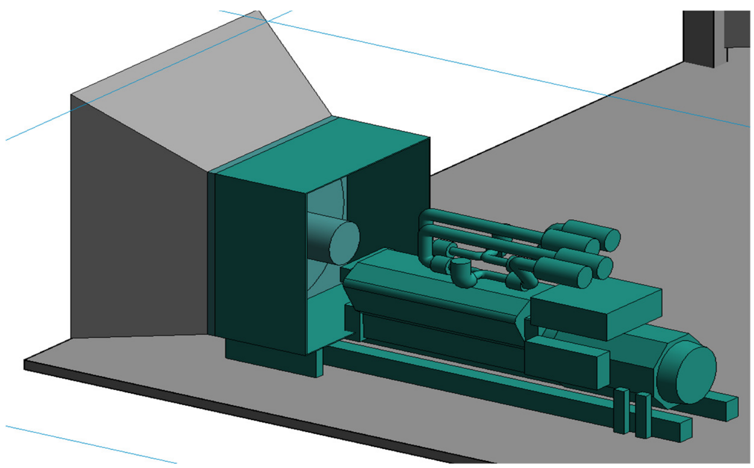
CFD modeling and analysis of DG room workflow starts with CAD model preparation, assigning materials and boundary conditions, setting up solver parameters for air flow and heat transfer and extracting key results.

CAD model

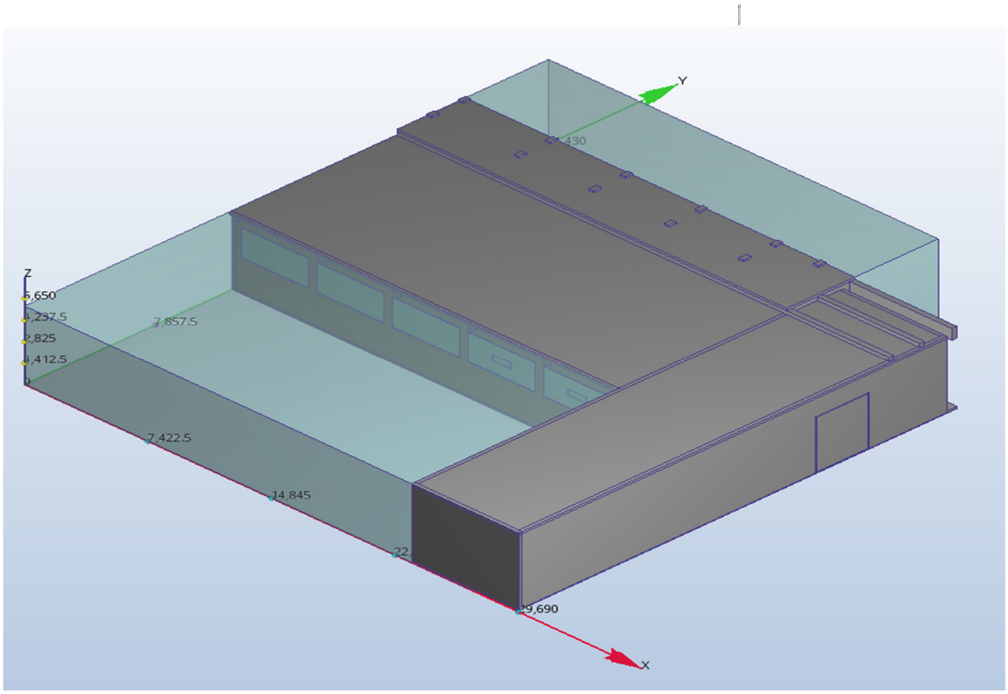
DG room geometry is created in Revit and geometrical details that are too small and not affecting air flow and heat transfer are removed or simplified. Especially DG engine set is simplified to make engine and alternator as volume heat generating elements. Next step is to add inlet and outlet extensions to model air flow realistically in and out of the DG room.



DG room Revit model



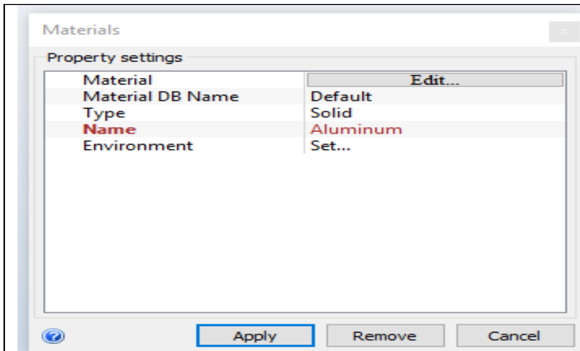
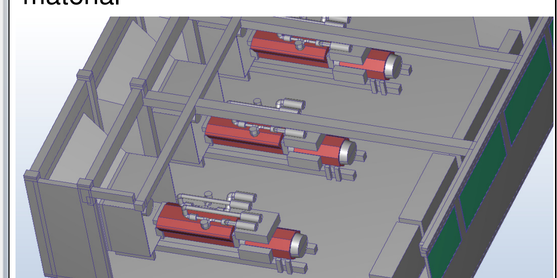
Simplified DG engine, alternator and other significant parts

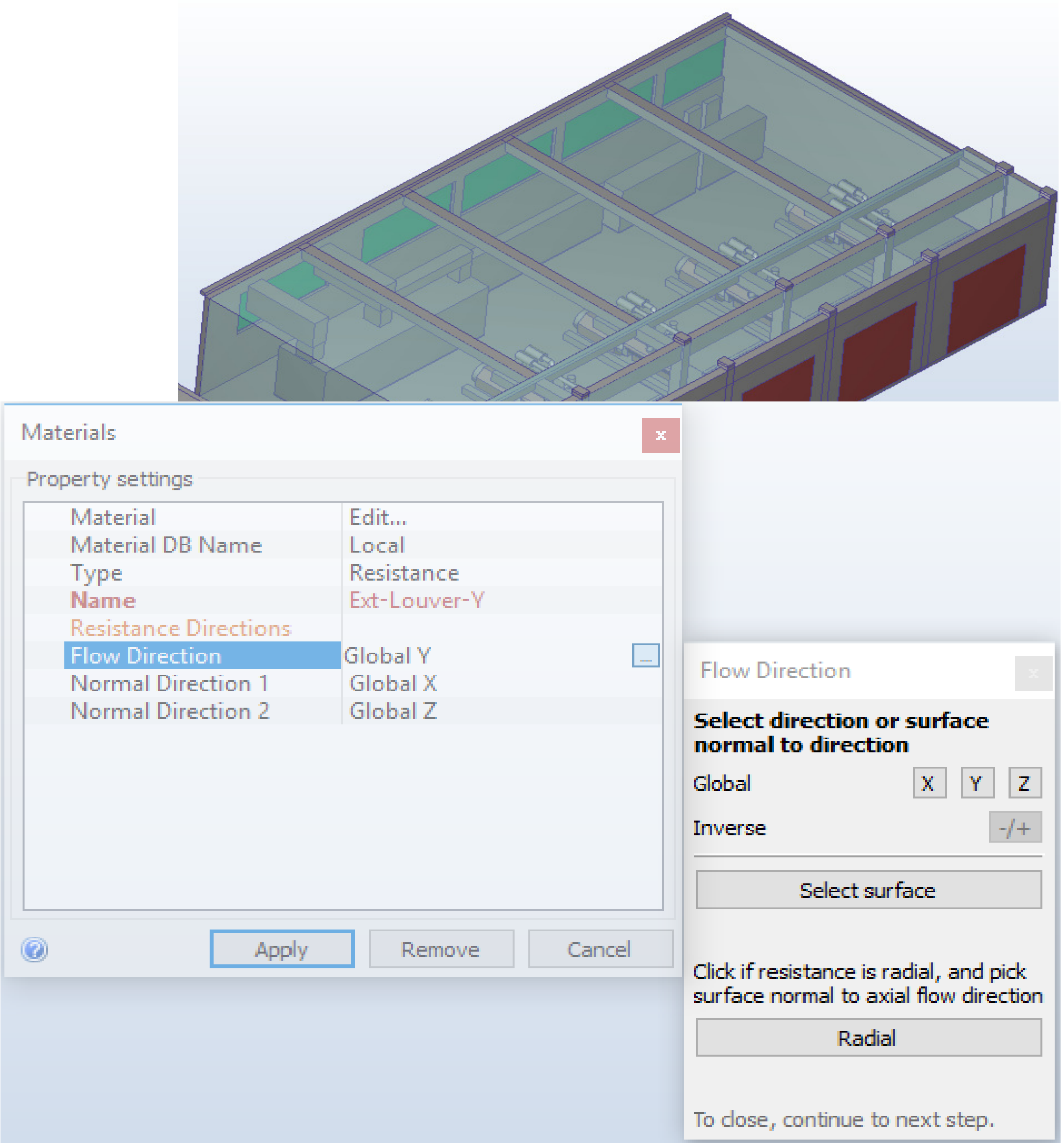


Inlet and outlet extensions created in CAD model

Materials assignment

Since heat transfer is by forced convection, air domain is assigned as air with fixed properties (density does not vary with temperature); inlet ventilation louvers, outlet louvers, radiator coil are assigned resistance material with free area ratio. Radiator fan is assigned with internal fan material. Engine and alternator are assigned with aluminum solid (high conducting material to transfer heat to ambient air). Other construction elements such as walls, panels and pipes are suppressed in the analysis.

<table>
<tr>
<td>
</td>
<td>Engine and alternator assigned Aluminum material
</td>
</tr>
</table>



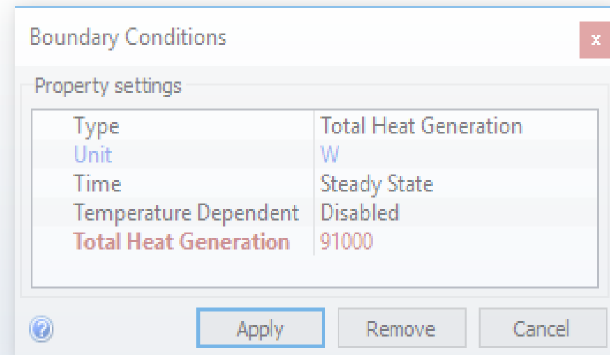
Outlet louvers assigned "Resistance" material with free area ratio of 0.55 in "Through-Flow" direction. In other directions "0" free area ratio is assigned to account for flow only in the specified direction.

Same material is assigned for inlet louvers also.

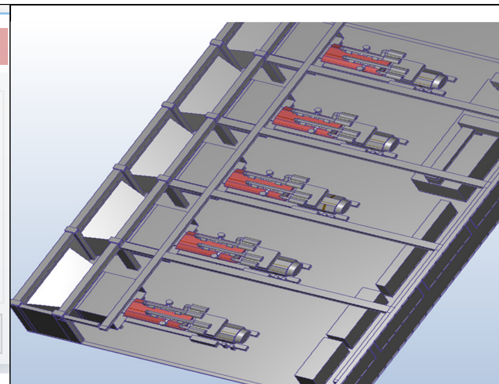
Radiator coil is also assigned "Resistance" material with free area ratio of 0.65 in "Through-Flow" direction.

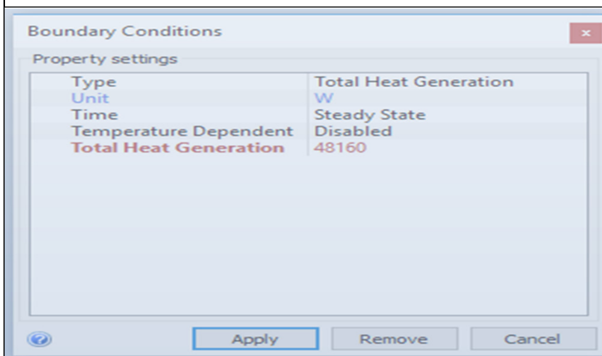
Boundary Conditions (BCs)

BCs for DG room include inlet flow boundary conditions (pressure and temperature), outlet flow boundary conditions (pressure), and inlet flow rates at combustion air inlets, total heat generation for engine, alternator and radiator coil.

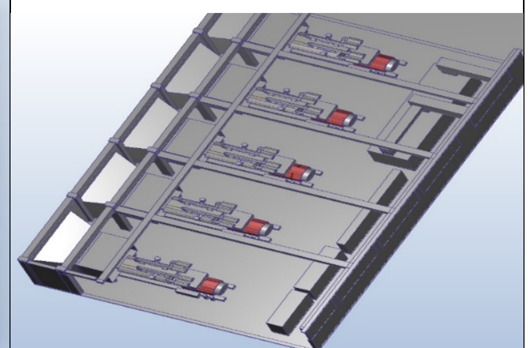


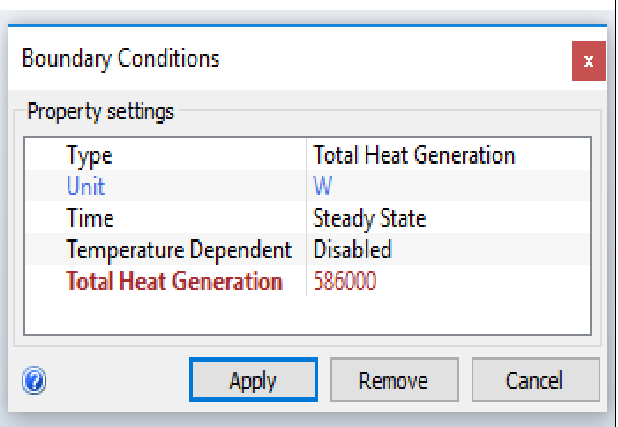
91 kW heat generation BC applied for Engine



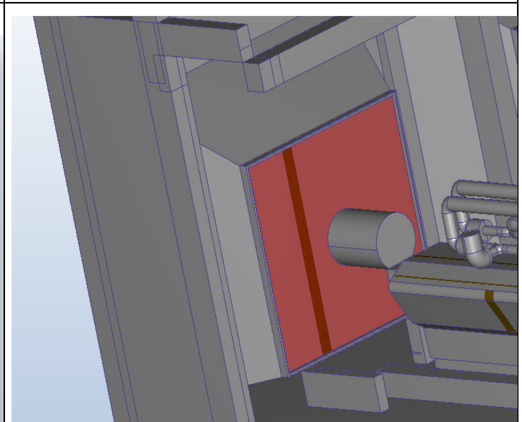


48.16 kW heat generation BC applied for Engine



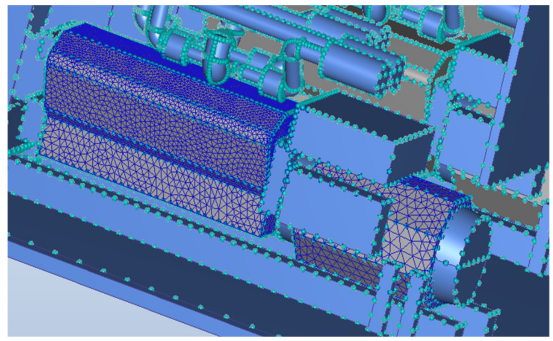
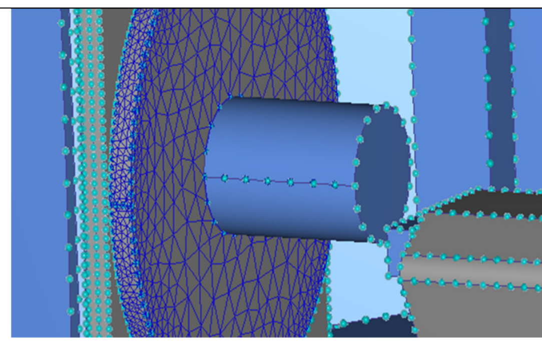
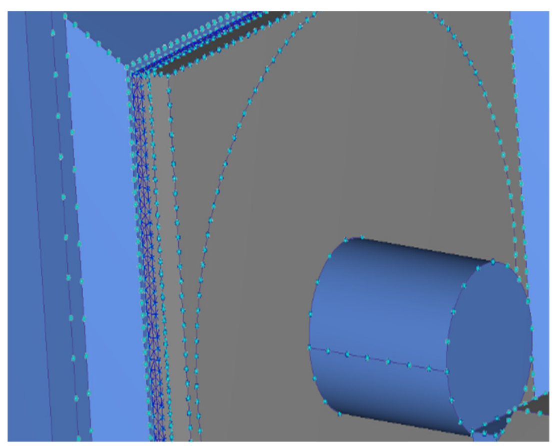


58.6 kW heat generation BC applied for Engine



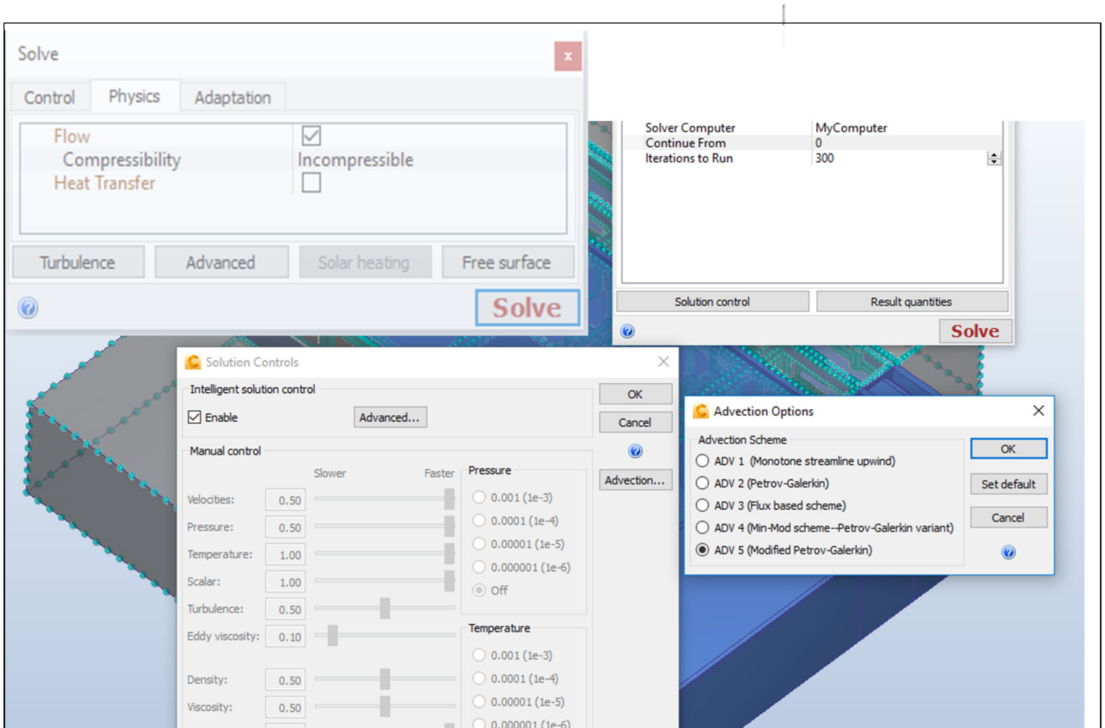
Meshing

It is important to mesh inlet and outlets with 4-5 elements to capture flow properly. Also radiator fan part to have uniform mesh with 4-5 elements. Fine mesh is applied on engine and alternator volume parts.

Fine mesh applied on engine and alternator parts to capture heat outflow and air flow over these parts.	
Uniform mesh to capture internal fan flow effects on radiator fan	
Uniform mesh to capture flow through radiator coil.	

Solver settings

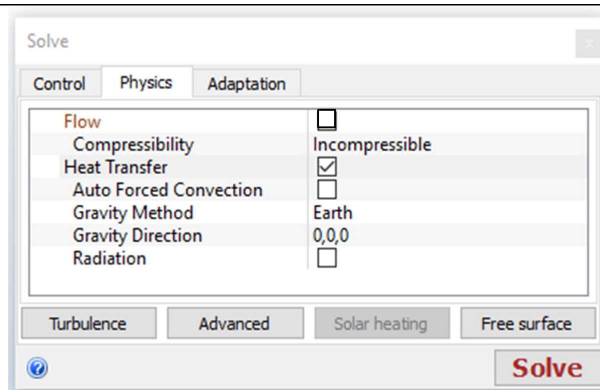
Heat transfer is by forced convection in this case. So flow and heat transfer can be decoupled during solution, i.e. solve for flow only first till flow convergence and then enable heat transfer till temperature convergence.



Solver settings for flow analysis (Intelligent solution control-On, Advection Scheme 5)

Switch off Flow
Switch on "Heat Transfer"

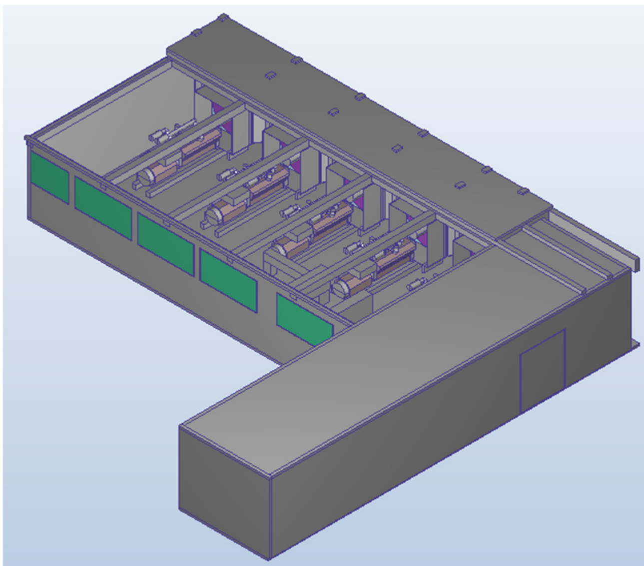
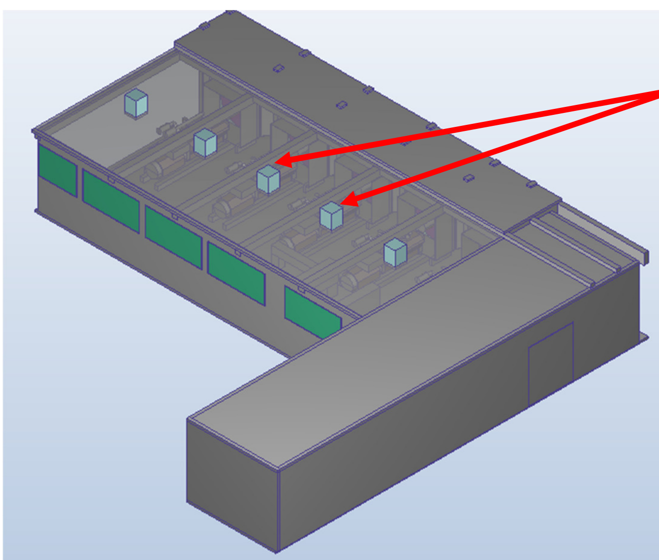
Continue to solve from previous iteration

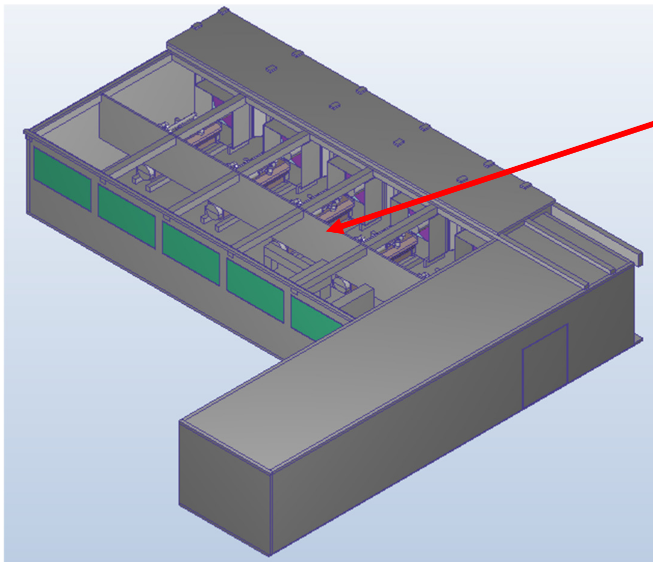


Solver settings for heat transfer analysis (Intelligent solution control-Off, Advection Scheme 5)

Key results for design performance –DG room ventilation

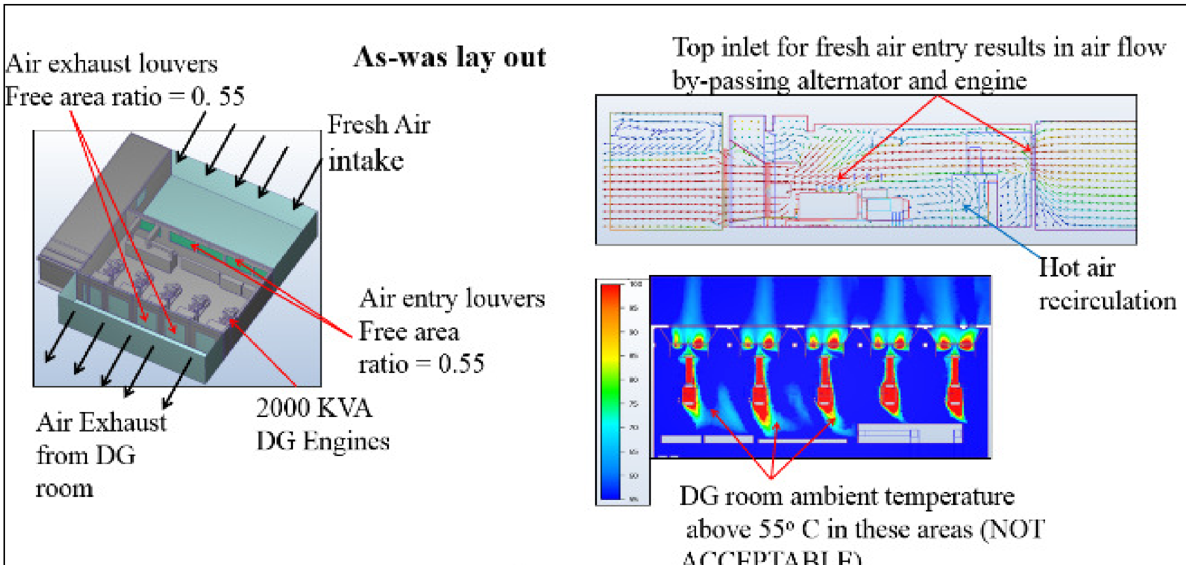
Useful results from CFD simulation for evaluating DG room ventilation performance are: Temperature results with "Global" option, Temperature results with "Planes" option and Velocity results with "Planes" with vector plot. In the following section, key results are highlighted for DG room ventilation design options to arrive at cost effective and innovative solution using Autodesk CFD.

	Original Design (as-was layout) This is the original design for the room.
	Design Option 1 In this option mechanical ventilation with additional fans were considered for cooling the DG directly by adding air circulation from the roof.

	Design Option 2 In this option an additional wall (baffle partition dropped from roof to a height of 2.1m from floor) is placed between intake louver and the DG containing/diverting airflow closer to the DG.

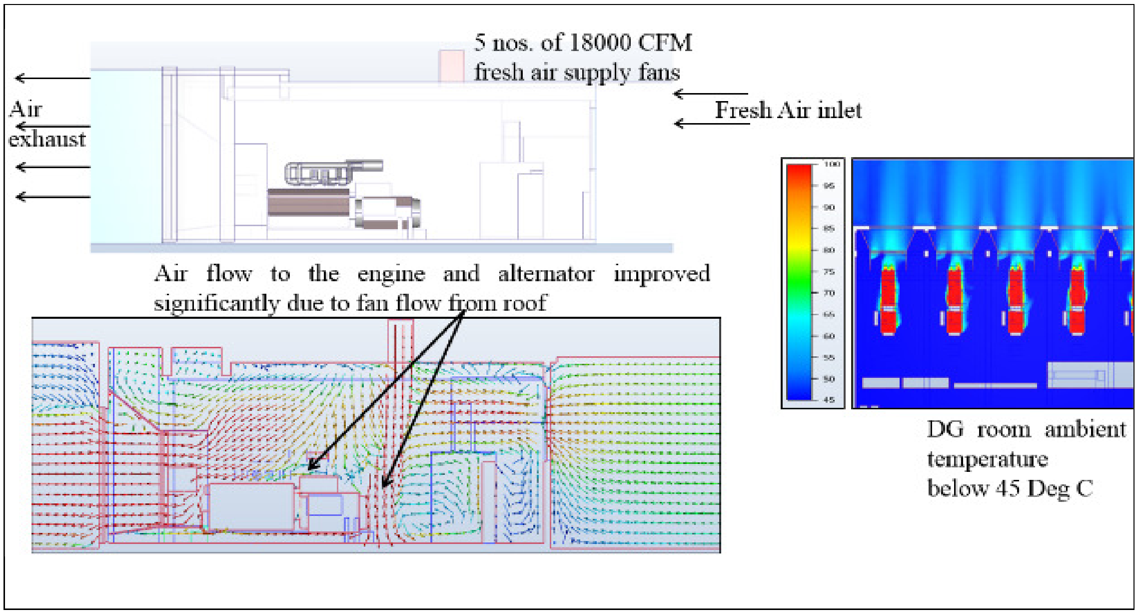
CFD results for Original Design (as-was layout)

CFD results for original design showed ventilation air bypassing alternator and engine. This resulted in hot-air recirculation and unacceptable room ambient temperature.



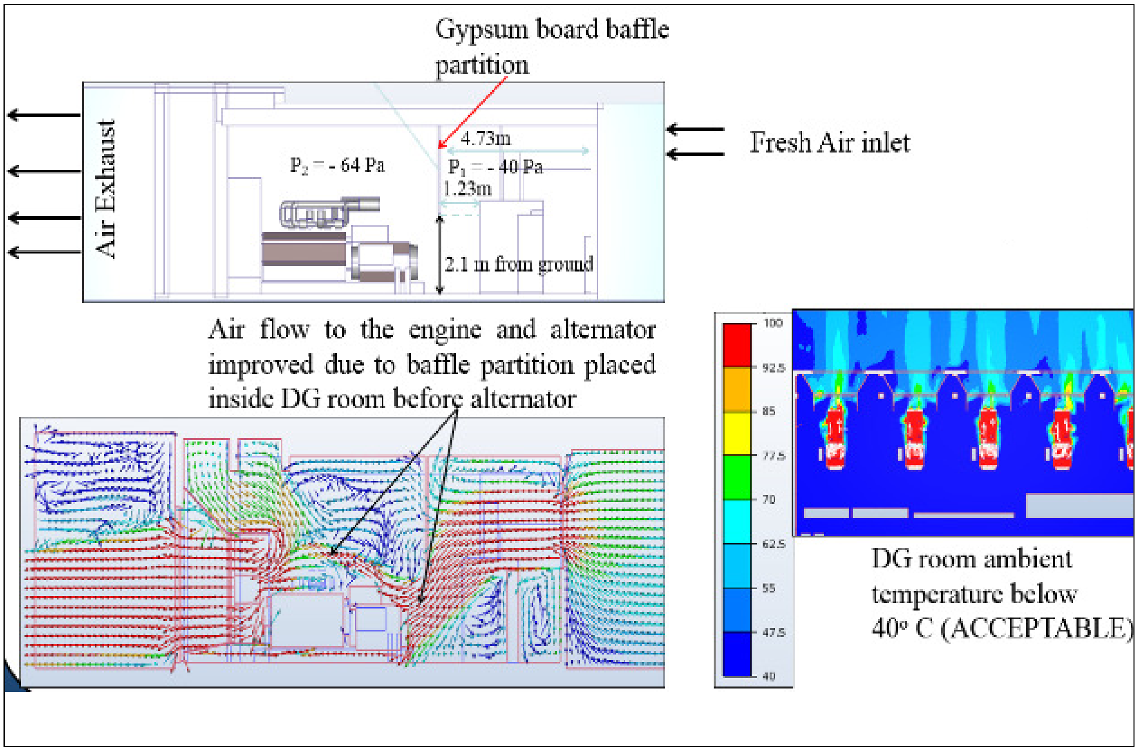
CFD results for Design Option 1

CFD results for this option with additional fans showed room ambient temperature is brought down to 45^0 C due to proper air flow over DG. But this solution is expensive…..



CFD results for Design Option 2

This option with additional wall (baffle partition) between the intake louver and the DG shows that air flow gets properly diverted over the alternator and engine surfaces and room temperature is brought down to 40^{0}C. And this solution is cost effective…..



From the above results, it is seen that design performance criteria for DG room ventilation is met cost effectively with Design Option 2.

- DG room ambient temperature is below 40^0 C
- Air flow is bottom to top due to baffle partition placed before the inlet louvers.

This solution is implemented and currently DG is very working effectively and providing continued backup power. Performance/ warranty issues are avoided with this design option.

Simulation strategies and techniques- Smoke visibility and extraction

Design goal is assessment of smoke exhaust design for safe evacuation and life safety in an Exhibit Hall of Convention Center. Initial design is based on traditional design approach using 12 ACH (Air Change per Hour).

Goal of CFD simulation is to obtain smoke visibility, smoke temperature and air flow fields using fire size instead of 12ACH approach and determine smoke exhaust capacity that meets the design performance criteria.

Design performance criteria:

- Smoke tenability limit is considered as 10m visibility based on BS PD 7974-6:2004, Annex G, Table G.1 for large enclosures and travel distances. [1]
- For atmosphere saturated with water vapour (such as in sprinklered fires), a tenability limit temperature of 60^0C is recommended for any exposure period up to 30 minutes as per BS PD 7974-6:2004, Table G.3 [1].
- Toxicity is deemed acceptable if visibility >10m

Smoke visibility

The Smoke visibility capability computes the distance a lighted sign can be seen from within a smoke-filled environment. The flow computed by Autodesk® CFD and a correlation developed by the United States NRC evaluate this distance based on the resultant air flow, location of the fire, and properties of the fire load.
Note: Smoke visibility is defined in terms of the furthest distance at which an object can be perceived within a smoke-filled environment. .
This information helps architectural designers determine the location and placement of exit signs in enclosed environments. As smoke fills a room, the visibility of signs that point to exits is vital for safe evacuation.

CAD model

The building considered for smoke extraction design is an exhibit hall of a large Convention Center. The building geometry is created in Revit and geometrical details that are too small and not affecting air flow and heat transfer are removed or simplified. Importantly fire part is created to act as fire source in the analysis. Inlet extensions are added to model air flow realistically through the doors wherein fresh air entry happens. Outlets consist of smoke extract fans placed on the roof and these outlets are also given sufficient extension to model air flow out of the building.

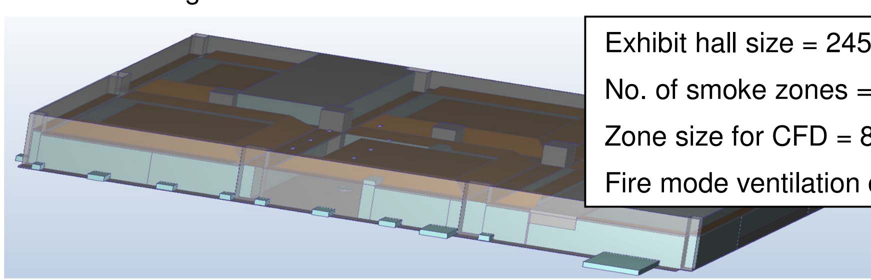


Exhibit hall size = 245L X160W X22.4H

No. of smoke zones = 10

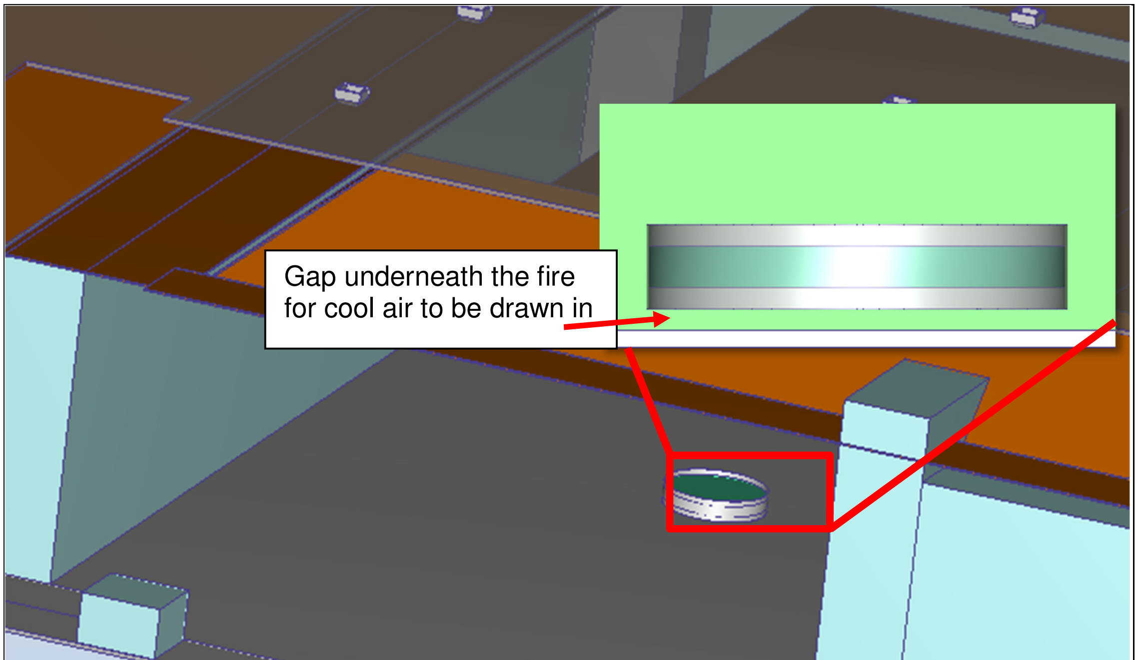
Zone size for CFD = 80m X 40 m

Fire mode ventilation design= 12 ACH

CAD model of Convention Center- Exhibit Hall

Materials assignment

All air volumes set to 'variable' rather than 'fixed', so that the density can change, allowing the air to rise as it is heated. Heat transfer happens by mixed convection mode due to mechanical ventilation and buoyancy. Important material assignment in smoke simulation is fire part. Fire part is assigned a “Resistance” material with “Through-Flow” constant and a high thermal conductivity. Other construction elements such as walls, floors, roofs and other elements not relevant to smoke simulation are suppressed in the analysis.

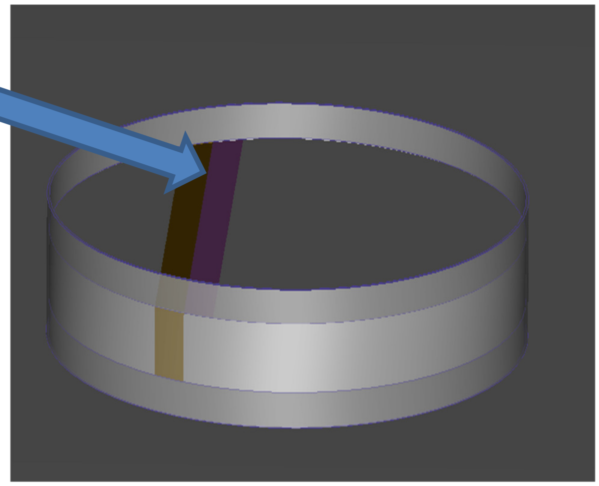


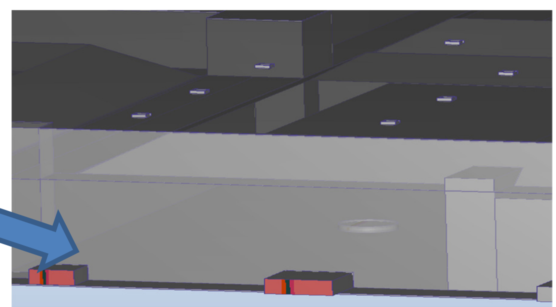
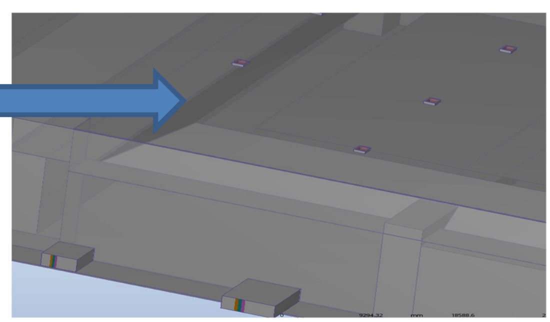
Close up view – Fire part

The fire part- a resistance with a 0.85 Free Area Ratio in flow direction and a Conductivity of 200 W/m-K). Solid around will be suppressed from the mesh and ensures that flow only enters and leaves through the bottom and top surfaces. The main flow direction is set to the vertical.

Boundary Conditions (BCs)

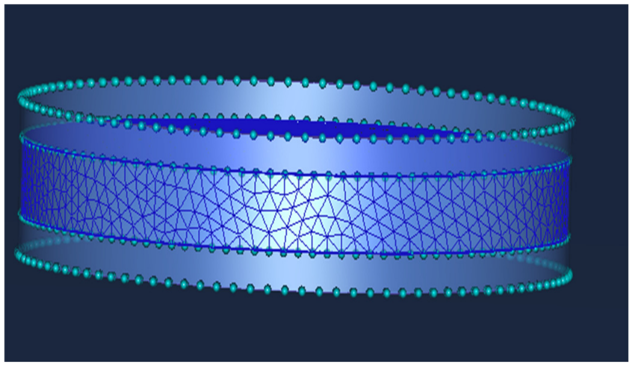
BCs for smoke simulation include inlet flow boundary conditions (pressure, temperature and scalar), outlet flow boundary conditions (volume flow rate). Fire source BC include total heat generation on the volume and scalar on the fire part top surface.

Fire part BC The fire has heat load applied to its entire volume. Scalar of 1 applied to its top surface Assign a **Scalar** boundary condition (value = 1) to indicate the location of the fire (smoke emission source). The scalar value represents the mass concentration of the combustion products throughout the analysis region.	

Inlet BC Scalar (0) Temperature (Celsius) Pressure (0 Pa)	
Outlet BC Volume Flow Rate (ft3/min)	

Meshing

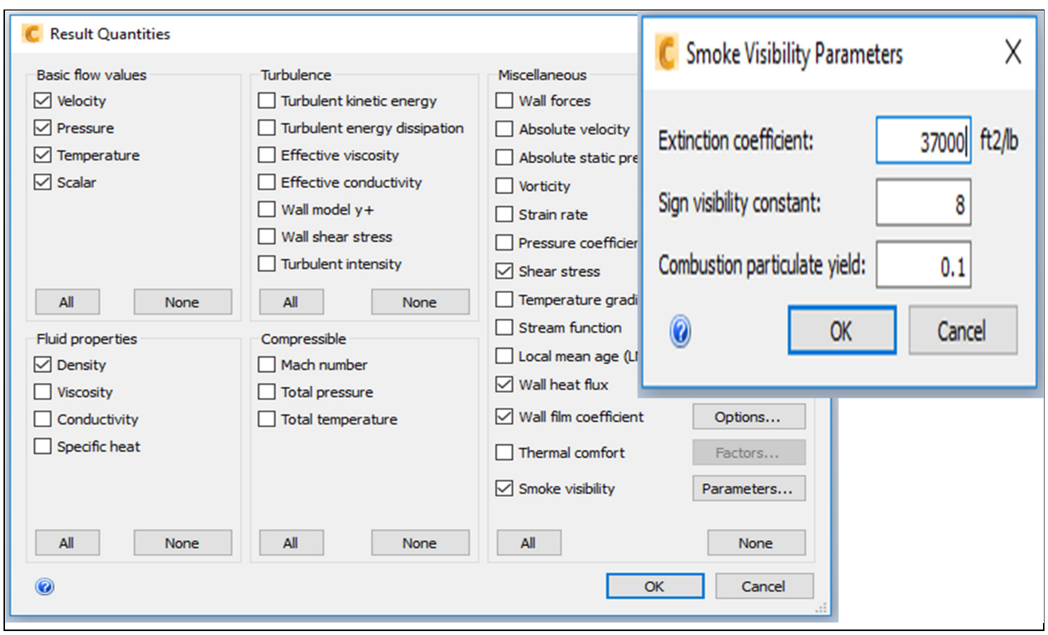
It is important to mesh fire part with uniform mesh at 4 - 5 elements to capture flow through it. Also fine mesh below and above the fire part to capture smoke plume and spread.



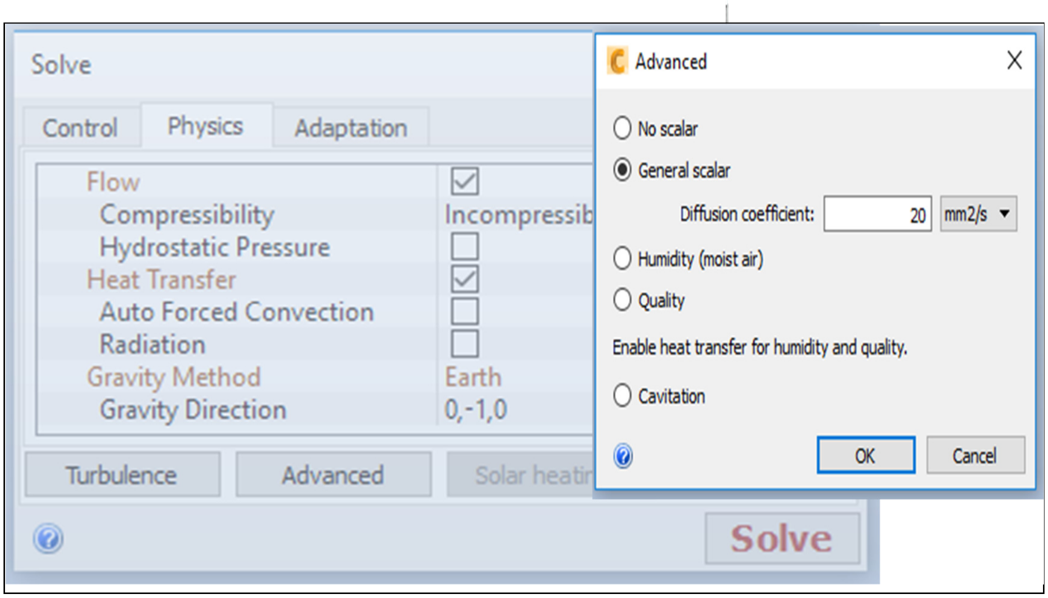
Uniform mesh on fire part

Solver settings

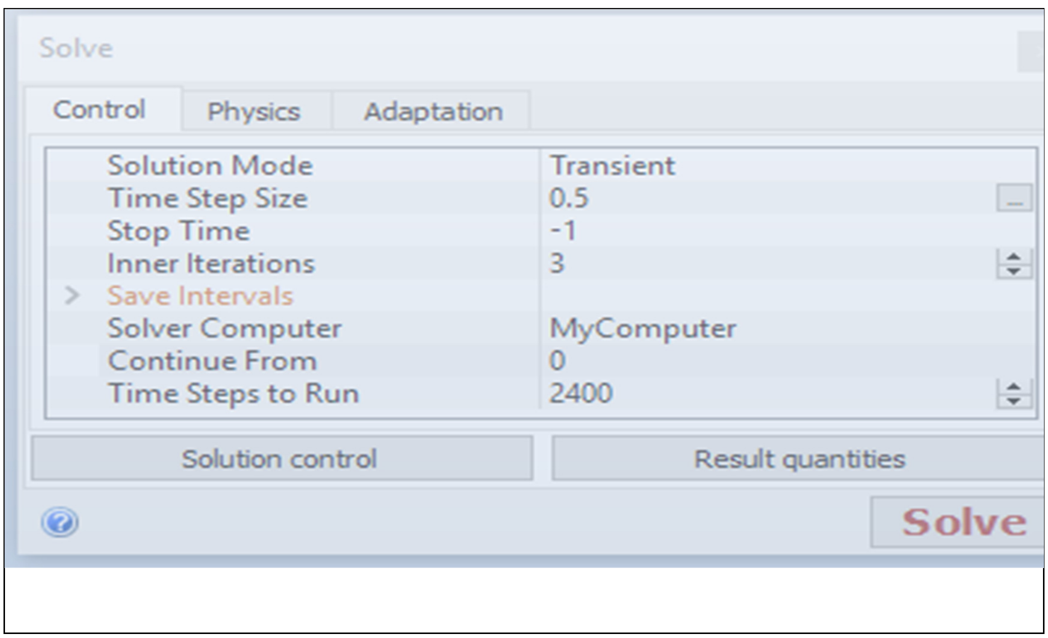
Heat transfer is by mixed convection. So flow and heat transfer are coupled during solution, i.e. solve for flow and heat transfer simultaneously. Transient solution with a small time step is used to achieve convergence. Important inputs are: smoke visibility parameters and scalar option.



Smoke Visibility parameters



Inputs on smoke as scalar



Transient analysis is specified with a time step of 0.5 s.

Key results for design performance –Smoke visibility and extraction

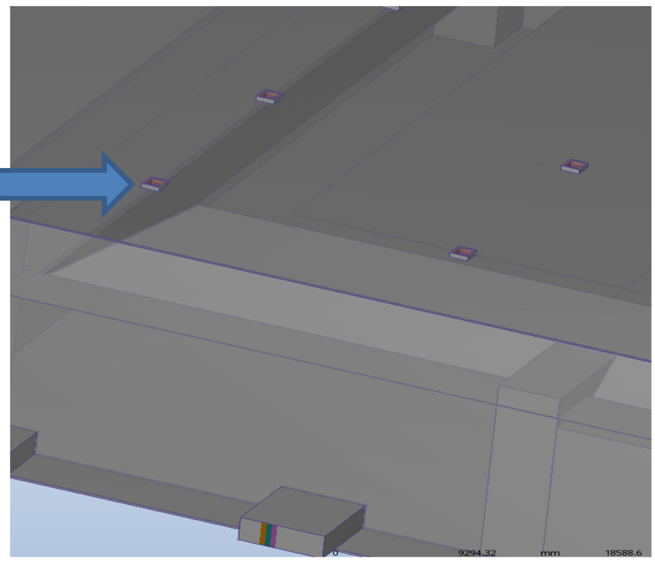
Smoke free space and tenable smoke temperature are the key results when we evaluate design performance of smoke exhaust system for life safety.

Three design options are considered to evaluate smoke exhaust performance as given below:

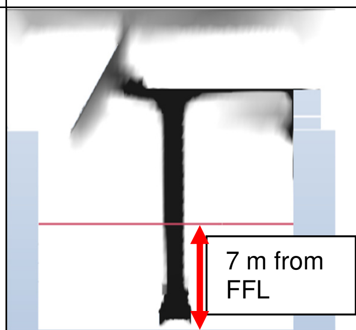
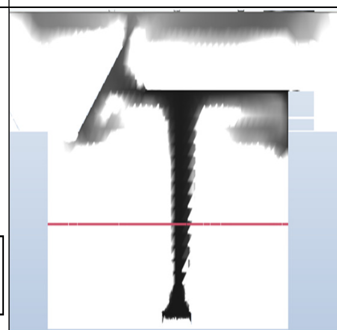
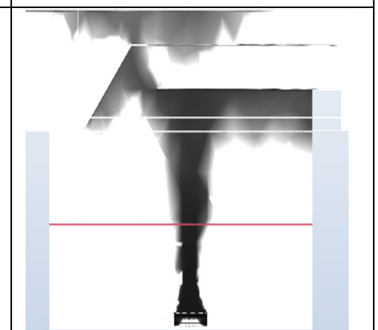
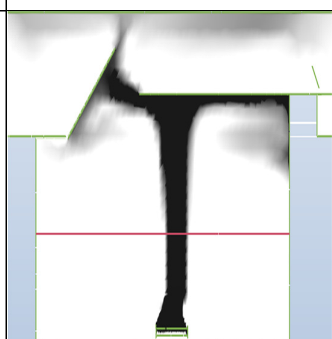
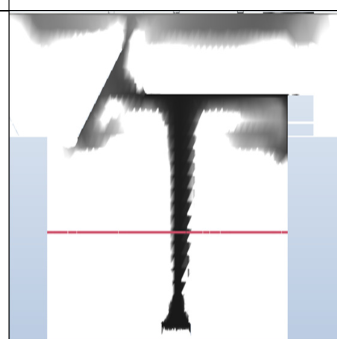
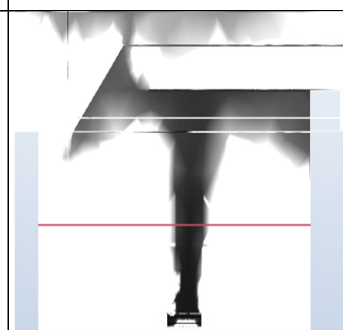
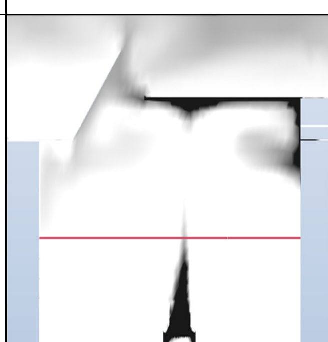
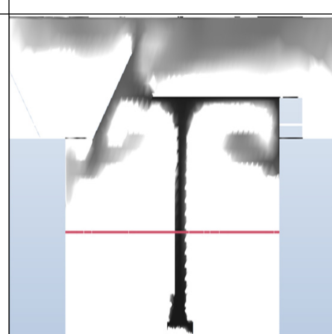
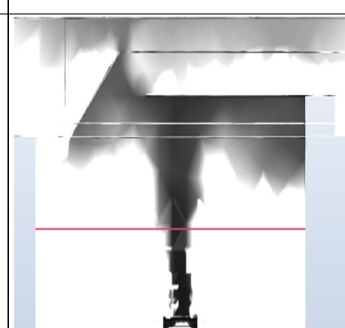
The three design options are based on smoke exhaust fan flow rate

Outlet BC:

- Initial Design: 20000 CFM
- Design Option 1: 15000 CFM
- Design Option 2 :12000 CFM

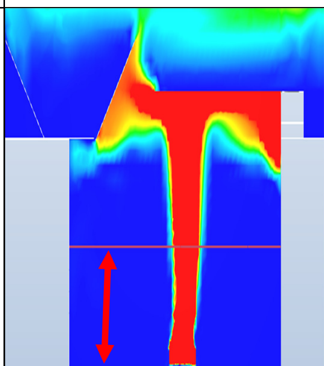
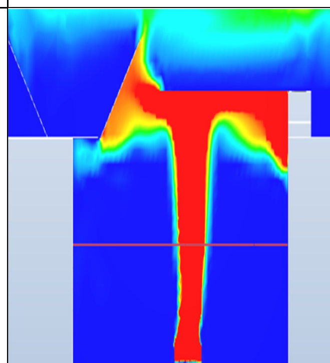
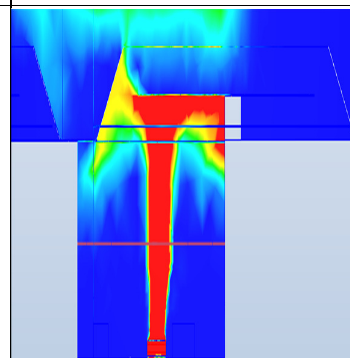
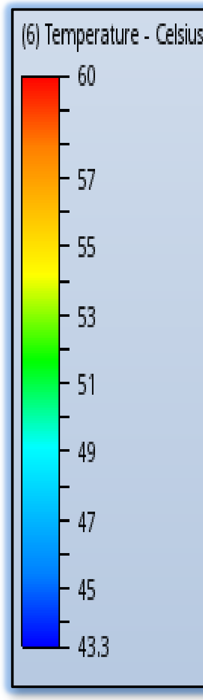
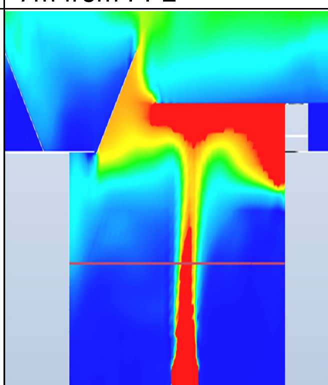
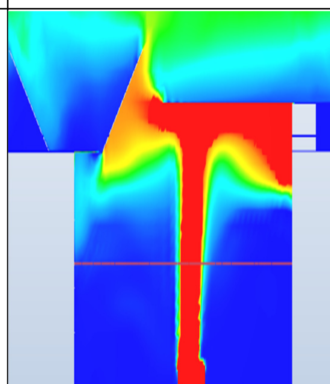
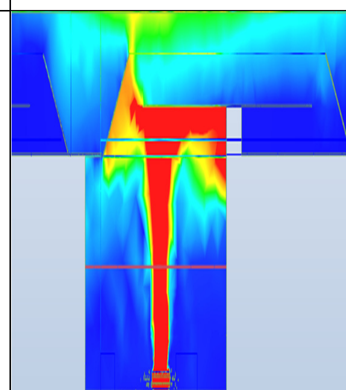


Smoke visibility results

Time (s)	Initial Design	Design Option 1	Design Option 2
60			
90			
180			

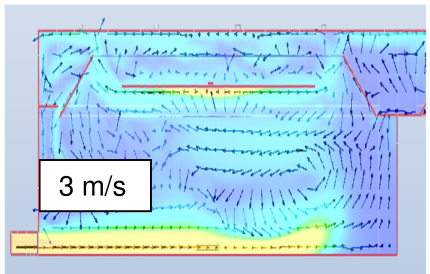
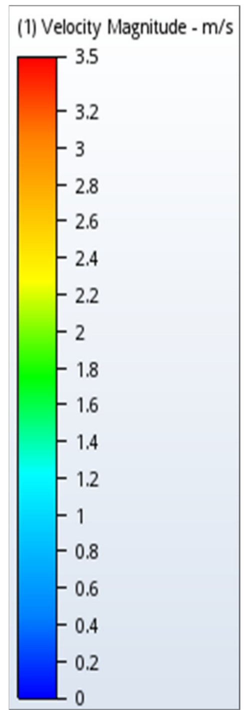
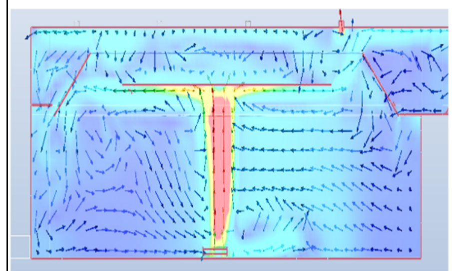
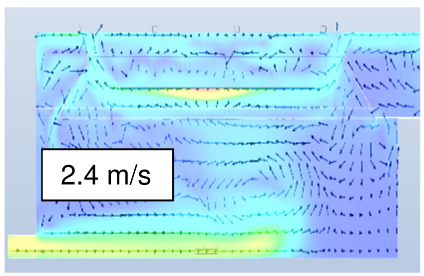
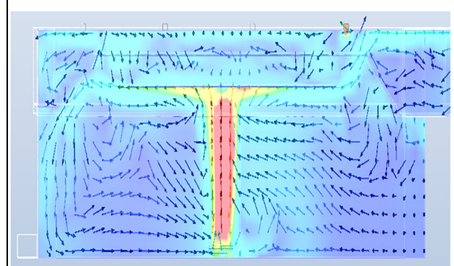
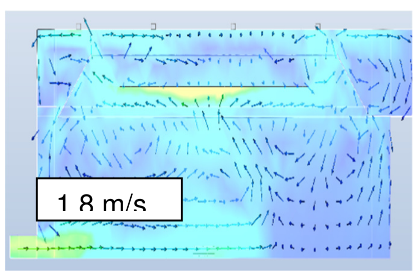
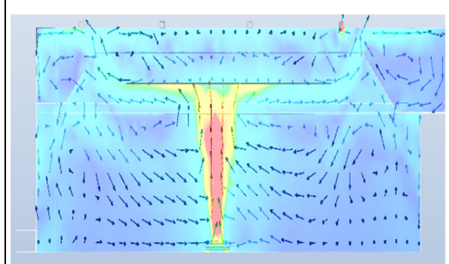
Occupied space is found to be smoke free (smoke free clear height of 7 m above the floor), thereby allowing safe evacuation of people in the event of fire.

Smoke temperature results

Time (s)	Initial Design	Design Option 1	Design Option 2	
90	 7m from FFL			
180				

Smoke temperature is below 60^{0}C and hence tenability limits for smoke temperature is not breached at 90 seconds and remains tenable up to 180 seconds from start of fire.

Air / Smoke flow field results

	Through door		Through fire
Initial Design			
Design Option 1			
Design Option 2			

Fresh air velocity contacting smoke plume has not exceeded 1.02 m/s all the cases. However, fresh air velocity for 20000 CFM and 15000 CFM is higher and hence stronger influence on smoke plume deflection due to stronger recirculating air around the plume. In the case of 12000 CFM case, smoke plume does not get deflected and rises as plume above the required smoke free clear height, hence 12000 CFM option is a better design.

Replacement/ makeup air velocity at entrance not exceed 2.4 m/s and hence does not hinder people escape through the entrance. The incoming airflow through escape doors should not exceed 5 m/s to permit unimpeded escape [2].

Design performance criteria for smoke visibility and smoke temperature is met for Design Option 2 (12000 CFM) and hence recommended.

AUTODESK. UNIVERSITY

Simulation strategies and techniques- Basement car park ventilation

Design goal is to evaluate ducted ventilation system for normal mode in a basement car park...

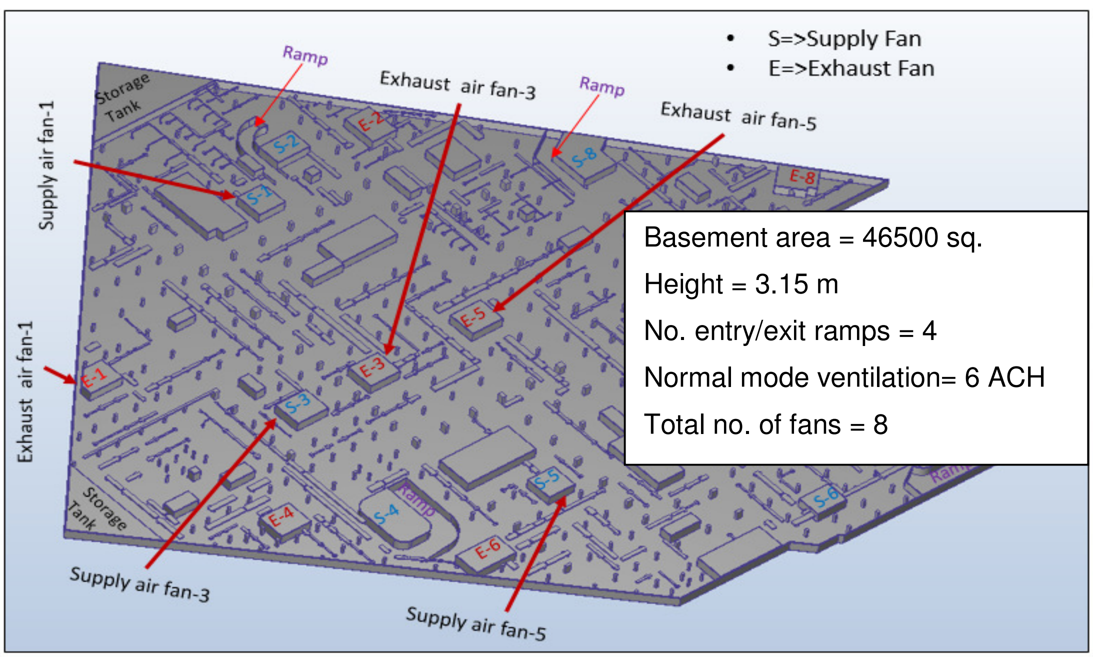
Goal of CFD simulation is to analyze airflow velocities and LMA (Local Mean Age), and recommend a better, cost effective design. Traditional design approach is to use 6ACH for normal mode ventilation. In this project, we have used an alternate approach to determine air supply design and flow rates to meet the required LMA values in the basement car park.

Design performance criteria:
The most appropriate results for analysing airflow are air velocities and LMA (Local Mean Age). In the LMA analysis, no recirculation is desired in any of the zones.

- The desired parameter for LMA in normal mode ventilation is 950 seconds [3]

Basement car park ventilation is achieved by either natural ventilation or mechanical ventilation. The goal in both cases is to ensure toxic gases emitted from vehicles is diluted/removed from the occupied levels in the basement so that human safety is ensured. In the event of fire, smoke is released from the burning vehicle and this smoke needs to be exhausted quickly so that it does not become a hazard to life. Either a ducted or ductless ventilation system is used for car parks. In ducted system, supply of makeup air into the basement and exhaust of contaminated air is taken care by designing and installing a network of ducts with supply/exhaust grills. In the project mentioned here, ducted system is considered for normal mode (daily emission) as well as emergency/ fire mode ventilation.

CAD model



Materials assignment

All air volumes set to 'fixed', so that the density cannot change with temperature. Only air flow is solved so all the solid elements such as walls, floors, roofs, service areas and other elements not relevant to air flow simulation are suppressed in the analysis.

Boundary Conditions (BCs)

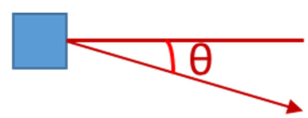
BCs for normal mode ventilation simulation include inlet flow boundary conditions (pressure), and outlet flow boundary conditions (volume flow rate).

Solver settings

Solver settings include flow physics with incompressible option and enabling LMA calculations in “Results Quantities” option

Key results for design performance –Basement car park ventilation

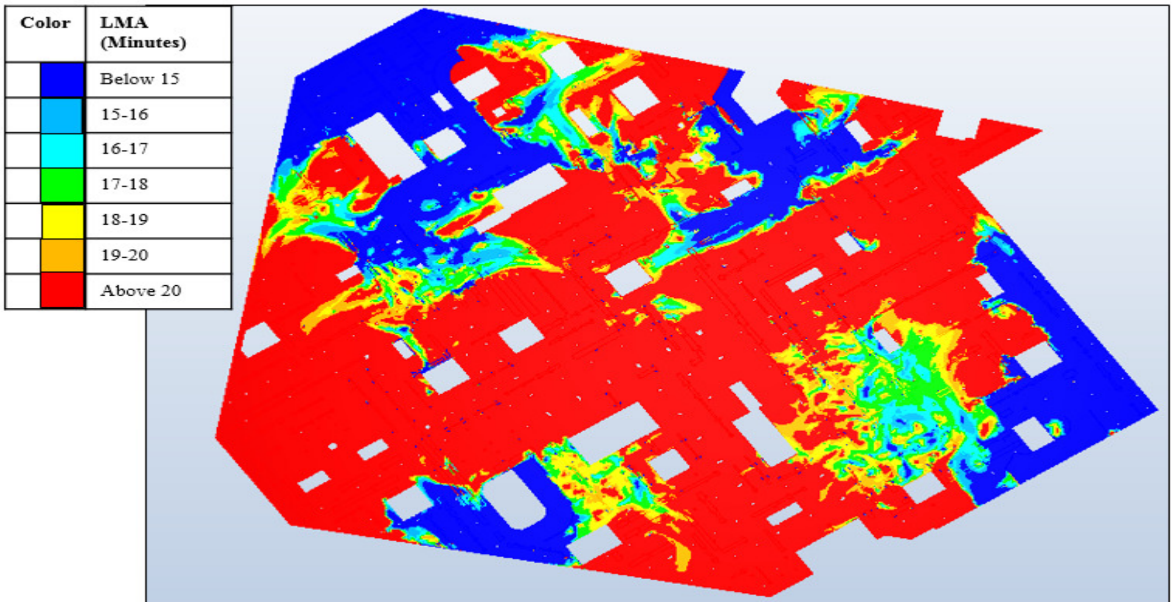
In order to evaluate ventilation effectiveness of ducted system, following design options were considered for CFD analysis



θ = Air throw angle

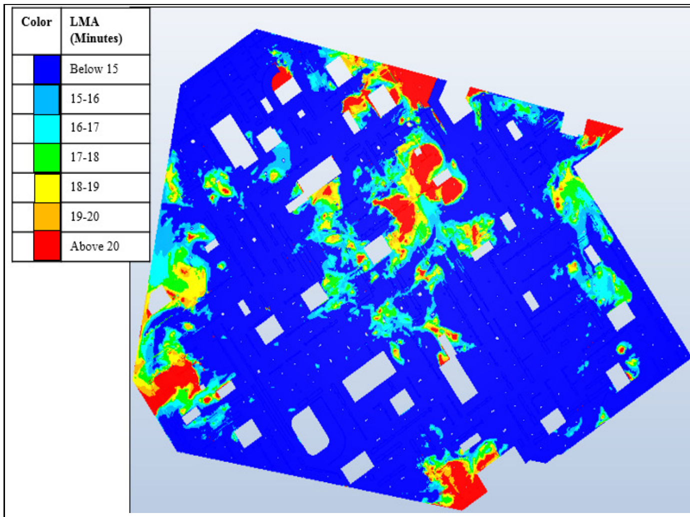
	Supply air flow direction (θ)	Exhaust air flow direction (θ)	Flow rate
Initial Design	0 degrees (horizontal)	0 degrees (horizontal)	Based on 6 ACH from 8 fans
Design Option 1	90 degrees (vertically downwards)	0 degrees (horizontal)	30% extra CFM from all fans (equivalent to 7.18 ACH)
Design Option 2	90 degrees (vertically downwards)	0 degrees (horizontal)	5 fans with 30% extra CFM, 3 fans with CFM doubled
Design Option 3	90 degrees (vertically downwards)	90 degrees (vertically downwards)	5 fans with 30% extra CFM, 3 fans with CFM doubled
Design Option 4	45 degrees	0 degrees (horizontal)	Flow rates doubled for all fans.

LMA values for Initial Design



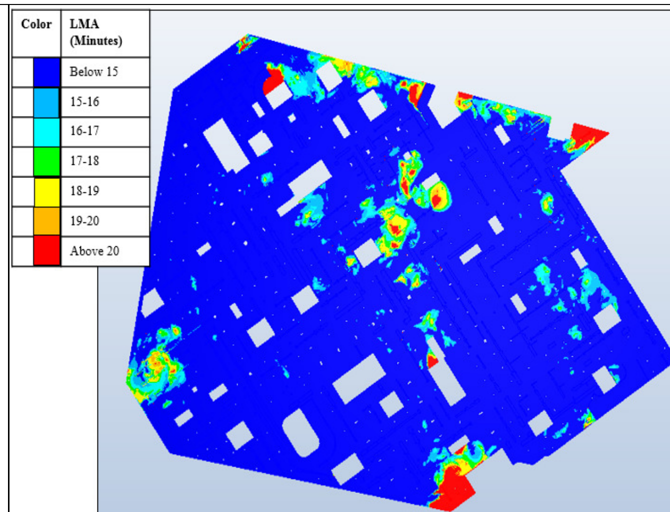
About 73% of area shows LMA values more than 15 minutes) at the occupied level (1.7m from the floor)
Normal mode (daily emission) ventilation- not satisfactory

LMA values for Design Options 1-4



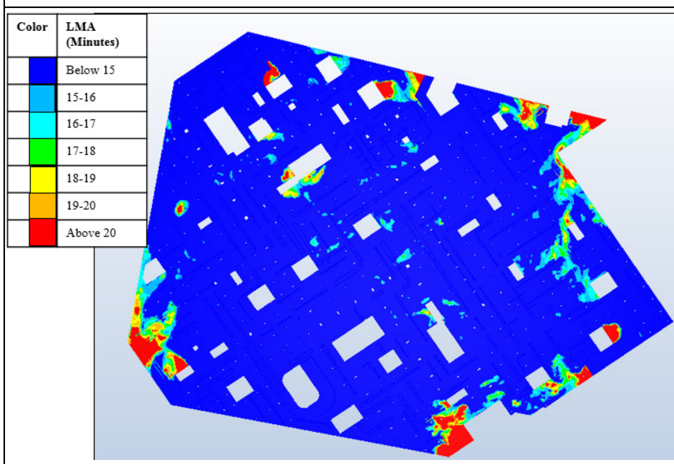
Design Option 1 :

About 73% of area shows LMA values below 15 minutes

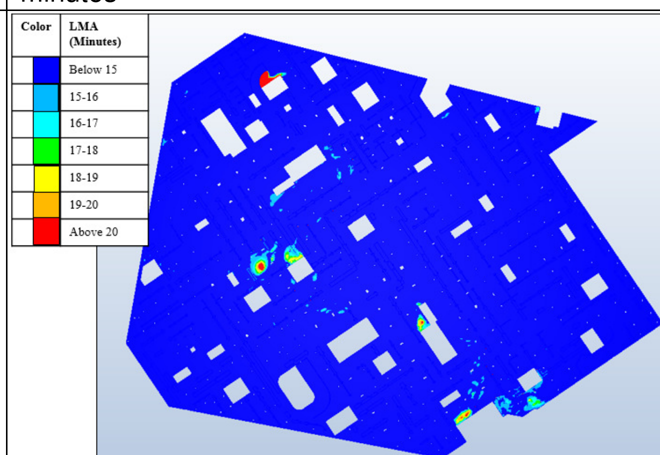


Design Option 2:

About 89% of area shows LMA values below 15 minutes



Design Option 3:
About 90% of area shows LMA values below 15 minutes



Design Option 4 :
supply direction -vertically downwards
About 98 % of areas having LMA value below 15 minutes

Design Option 4 is a better design but ventilation with LMA values less than 15 minutes is achieved by increase in fan capacity i.e. fan capacity at 12 ACH.

However, Design Option 3 provides a cost effective optimum design where acceptable ventilation is achieved by 78% fan capacity by changing flow direction vertically down wards for both supply and exhaust, thereby saving cost of fans by 22%. Hence Design Option 3 meets the design performance criteria and it is cost effective.

Simulation strategies and techniques – Rain water flow simulation

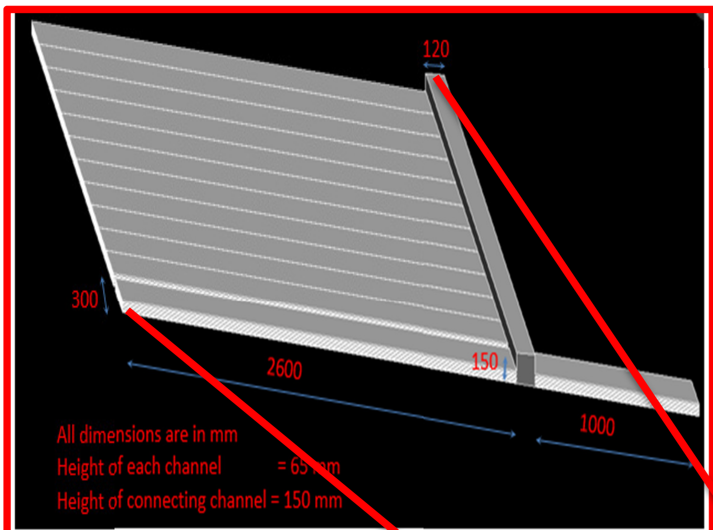
Due to heavy rain, rain water drainage system on the roof of an airport was found to be ineffective and resulted in water leakage into passenger occupied space in the airport.

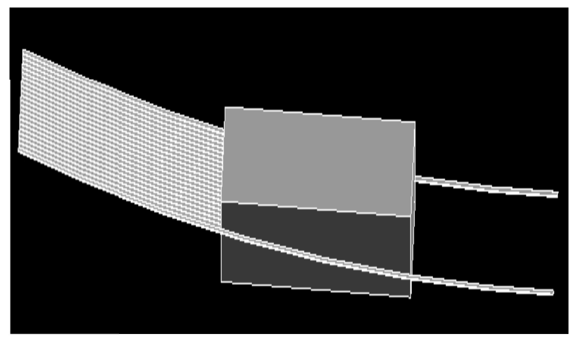
Goal of simulation is to predict water levels at Skylight obstructions in the roof water drainage channels and recommend corrective solution. Rain water flow is simulated using "Free Surface Flow Modelling" capability in Autodesk CFD.

Design performance criteria:

- Water level in the channel upstream Skylights should be below 65mm i.e. below the standing seam (which is 65mm tall) of channels.

CAD model





Length of roof (ridge to gutter) = 75m

Length of channel for CFD =2.6 m

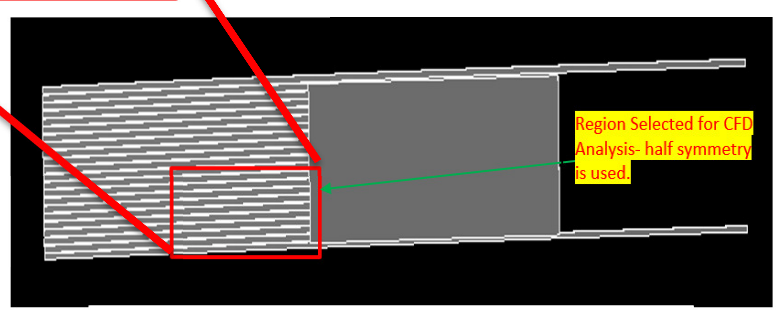
No. of channels per skylight = 24

Channel width = 300 mm

Height of each channel = 65m

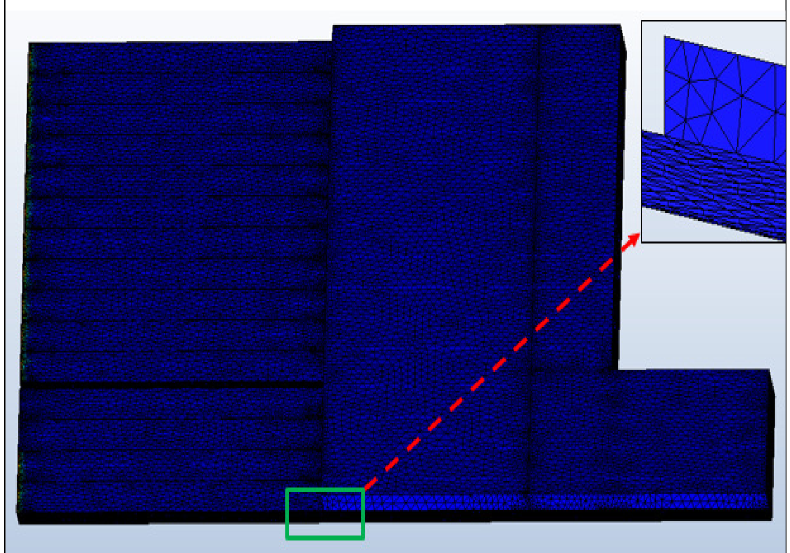
Rain intensity = 205 mm/hr.

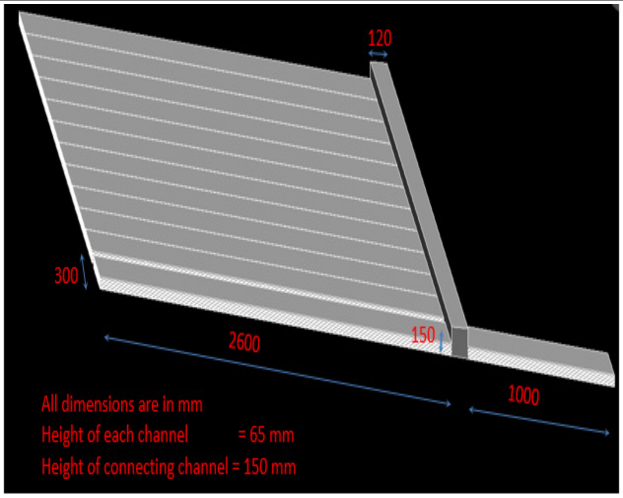
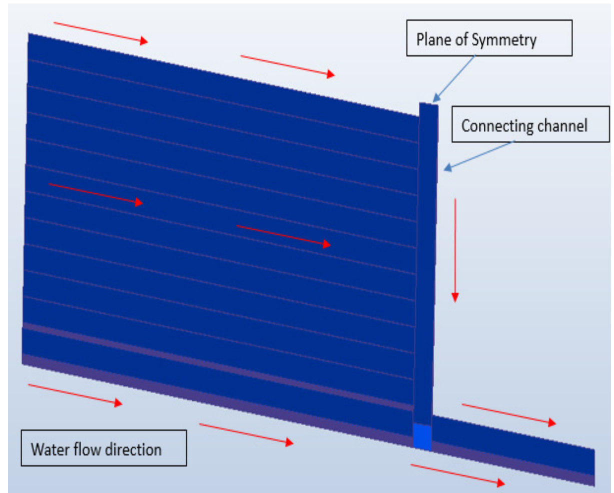
Flow through each channel = 0.8 lps

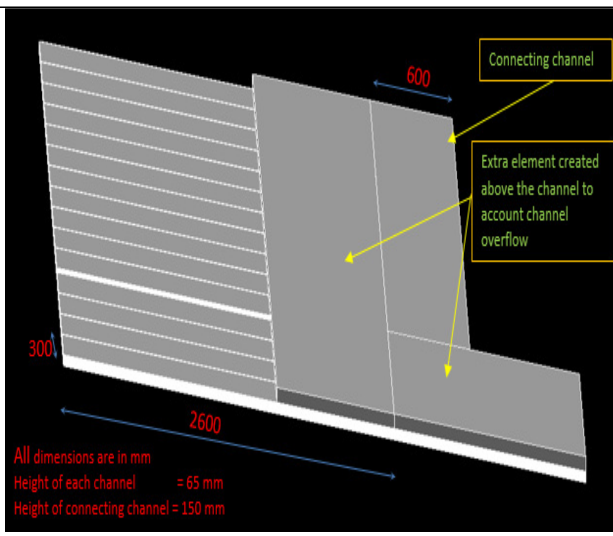
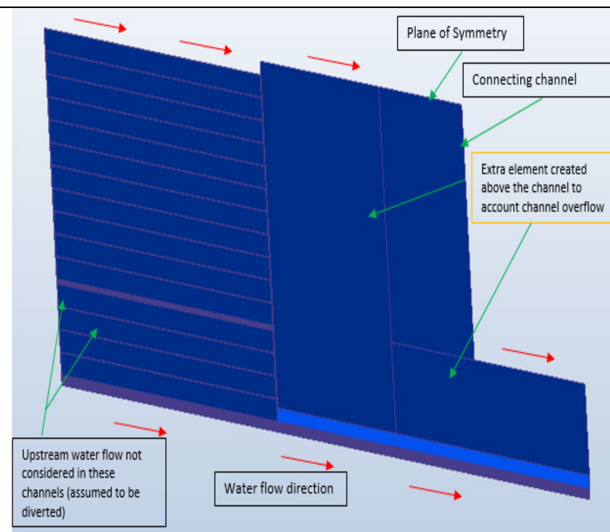


Free surface flow modelling

Free surface flow modeling involves assignment of liquid (water) to all parts.

Material Properties: Water: Fixed Quantity Boundary conditions: Inlet: Velocity-0.041 m/s at each strip inlet. Outlet: Pressure 0 Pa (Gauge). Plane of Symmetry: Slip/Symmetry boundary condition. Solution Parameters: (As per Autodesk simulation CFD software) Flow, Transient: ON Free Surface: ON (Earth gravity direction given) Heat Transfer, Gravity: Switched OFF Radiation : Switched OFF Turbulence: Switched ON – K-e Turbulence	

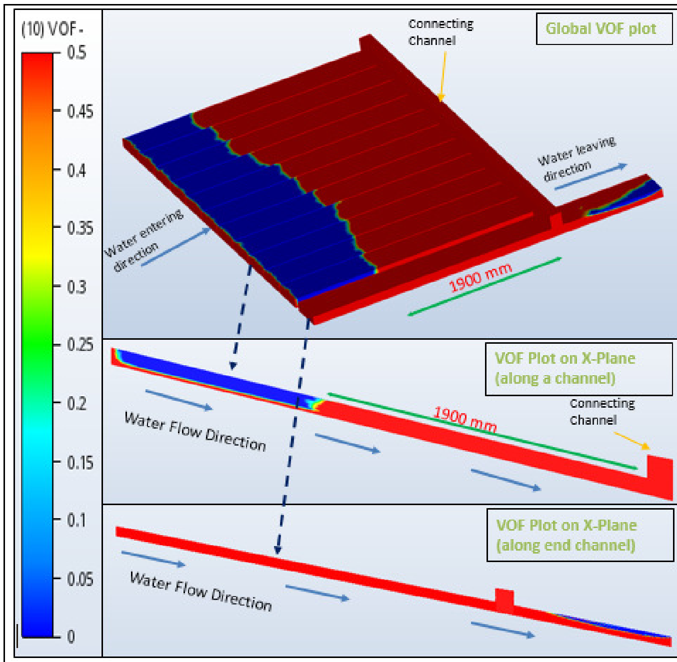
 CAD model- Initial Design Connecting channel width = 120 mm	 CFD model- Initial Design

 CAD model- Final Design Connecting channel width = 600 mm (arrived at after CFD analysis for increasing width from 120 mm to 600 mm incrementally)	 CFD model- Final Design

Key results for design performance –Rain water flow simulation

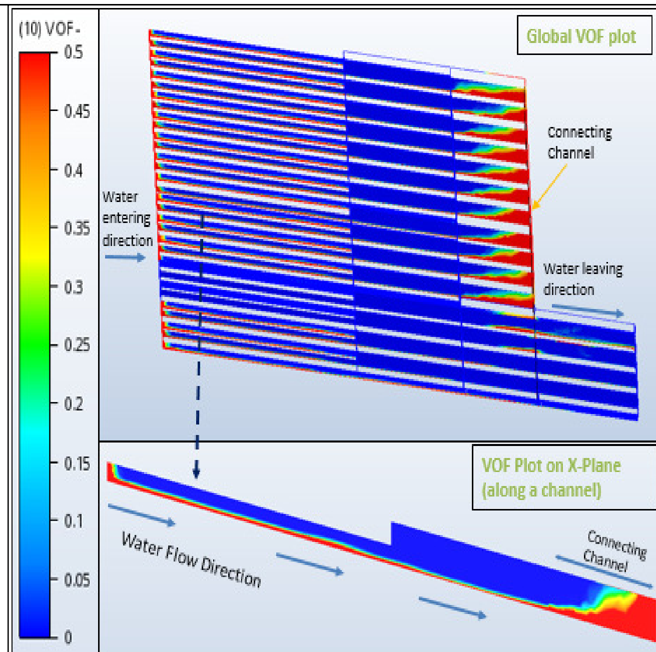
VOF results for rain water level- Initial Design and Final Design

Volume of Fluid (VOF) Plots shows the flow or level of fluid. VOF = 0.5 is the boundary between the water and air (Free Surface). The red color in the following plots indicates the portion of channel filled with water.



Results - Initial Design

Water level in all channels upstream skylight is above 65 mm and leak can take place from standing seam gaps.
The level of water in the downstream side of the connecting channel (critical area) where it meets the long channel is above 65 mm (above the height of channel), hence water can leak.



Results - Final Design

Water level in all channels upstream skylight (except connecting channel) is below 65 mm and hence leak cannot take place from standing seam gaps.

The level of water in the downstream side of the connecting channel (critical area) where it meets the long channel is 45 mm (well below the height of channel), hence water cannot leak.

Based on CFD results for VOF from free surface flow simulation, it is concluded that design performance criteria is met with 600mm wide connecting channel upstream the Skylights.

Advantages and limitations of using Autodesk CFD for MEP design

Advantages

- Easy to use
- Better designs
- Basis for innovative design
- Cost effective designs
- Avoidance of performance /warranty issues

Limitations

- As good as assumptions
- Computationally intensive (better accuracy requires more time)
- Depends on limited test data for validation
- Relevant test data not available for real life scenarios
- Requires knowledge and experience

References

[1] BS PD 7974-6:2004, *The application of fire safety engineering principles to fire safety design of buildings- Part 6: Human factors: Life safety strategies-Occupant evacuation, behaviour and condition (Sub-system 6), Clause 9.5,* **British Standard**

[2] BS PD 7974-2:2002, *Application of fire safety engineering principles to fire safety design of buildings- Part 2: Spread of smoke and toxic gases within and beyond the enclosure of origin (Sub-system 2)), Clause 9.5,* **British Standard**

[3[A. Senveli, T. Dizman, A. Celen, D. Bilge, A.S Dalkilic , S. Wongwises*, CFD Analysis of Smoke and Temperature Control System of an Indoor Parking lot with Jet Fans***, Journal of Thermal Engineering, Vol.1, (2) (2015) 116-130.**