AUTODESK UNIVERSITY

BES502380

Better HVAC Designs Using Autodesk CFD for Airport Expansion Project

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

Learning Objectives

- Implement AEC Simulation best practices in Autodesk CFD
- Apply relevant modelling strategies for airflow and heat transfer
- Assess Autodesk CFD results for better designs
- Identify opportunities for design optimization

Description

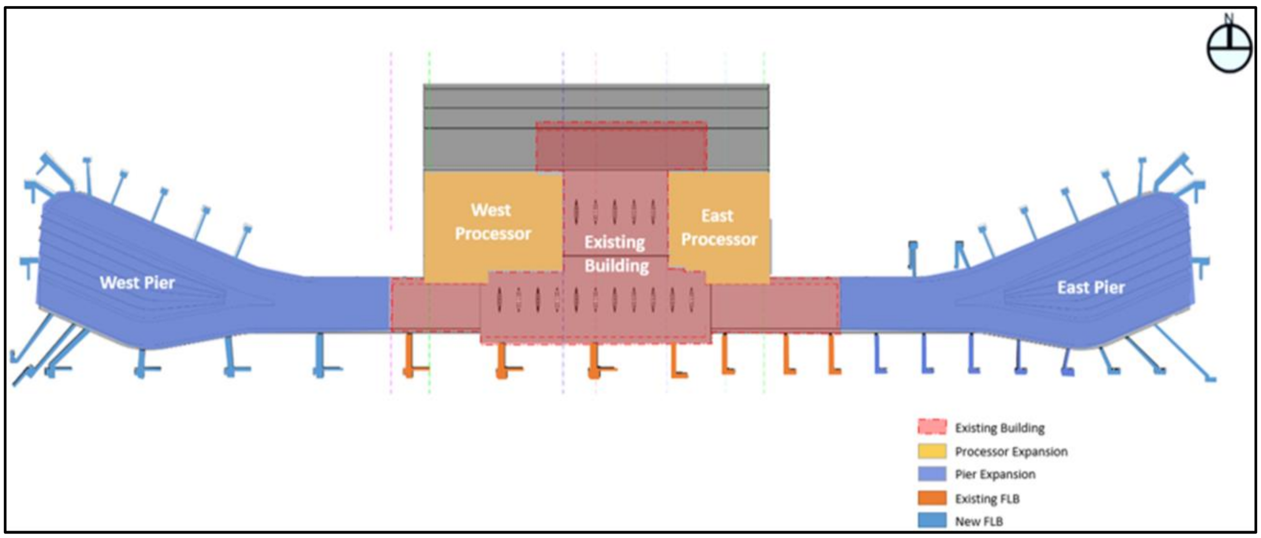
Passenger Terminal Building (PTB) is a very important feature of a modern, international airport complex. Expansion of airport is aimed at design and construction of additional buildings and facilities to service growing international and domestic passenger traffic. Baggage reclaim hall on the arrival side and check-in hall on the departure side are large volume spaces in PTB, requiring proper distribution of conditioned air for adequate human comfort. In this class, how Autodesk CFD is used to evaluate air-conditioning design is presented. Importance of airflow and temperature distribution from CFD results is highlighted to ascertain indoor air quality and thermal environment. Strategies for modeling airflow and heat transfer are detailed out with best practices for flow and heat load characterization. Challenges of unacceptable thermal environment (hot spots) are addressed through design changes and optimization of supply flow rates and discharge angles, resulting in better designs.

Speaker(s)

Dr. Munirajulu. M, Bachelor of Technology (Hons.) and Ph.D. from IIT, Kharagpur, India, has more than 26 years of industry experience using CFD technology for 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 factories 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, thermal comfort in airports, 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, 2020 US and AU India 2019,2021.

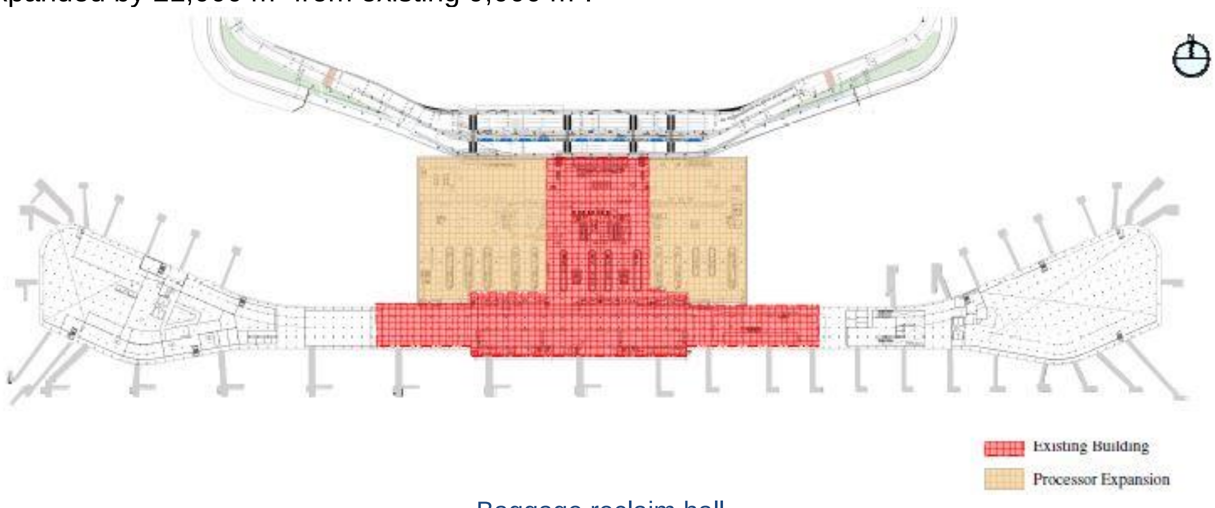
Introduction

The international airport is designed as international and domestic hub with ultimate capacity to carry 34 million passengers per annum from existing 12 million passengers per annum. Original building is T-shaped, and expansion gives it shape of a bird with wings spread. The airport complex consists of many different buildings and facilities such as runways, aprons, passenger terminal buildings, cargo terminal, air traffic control towers and aircraft hangers. Our focus in this case study is passenger terminal building, PTB in short as shown in the image below. The passenger terminal building has a total floor area of 120,000 m^2 and consists of functional areas over seven (7) levels and being expanded to a total floor area of 370,000 m^2.



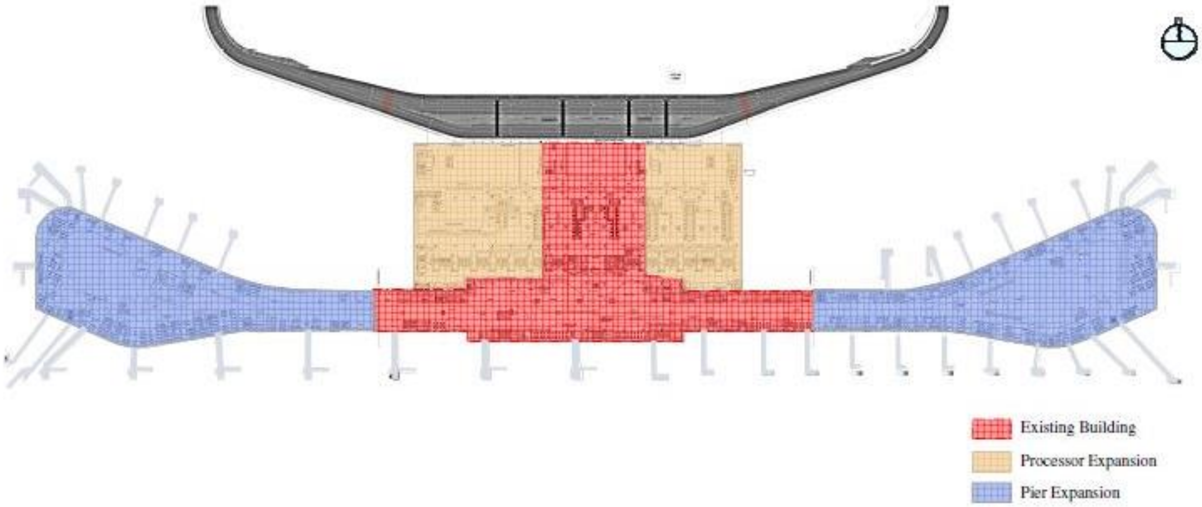
Passenger Terminal Building (PTB)

Within the passenger terminal building, baggage reclaim area in the arrivals section is being expanded by 21,000 m^2 from existing 9,000 m^2.



Baggage reclaim hall

Similarly, check-in hall area in the departures section is being expanded by 21,000 m^2 from existing 9,000 m^2.



Check-in hall

Air-Conditioning Design Basis

Both the check-in hall and baggage reclaim are passenger circulation areas with large spaces and environmental conditions are critical for passenger comfort. Air-conditioning design and its verification by CFD simulation is important to ensure comfortable environment for passengers. HVAC design conditions are based on specified outdoor condition for summer and indoor condition for passenger comfort. For CFD simulation, these design parameters along with external and internal heat gains including solar and non-solar sources are the input data for the models.

Air-Conditioning System

Air-conditioning is a chilled water-based system with centralised chilled water supply. Conditioned air distribution takes place through floor-mounted air-handling units, binnacles, and air terminal devices. Jet nozzles and slot diffusers are used as air terminal devices for check-in hall area. Drum louvers are used as air terminal devices for baggage reclaim area.

With this introduction about the project, let us move onto the key objectives.

.

Simulation best practices in Autodesk CFD

Component characterization

First and foremost, key feature for a successful CFD simulation is how we characterise or idealise building components and other geometrical features for CFD modelling. Large spaces and many components with tiny details as common in AEC applications need to be simplified through characterization. Complex features can be simplified through characterization to reduce simulation run time significantly which depends on mesh cell count in CFD. The following are characterized in CFD modeling for airport air-conditioning analysis:

- People/occupancy
- Air terminal devices such as slot diffusers

It is important to keep the “big picture” of simulation goal in mind during characterization process and how this process can reduce simulation complexity. Important considerations are:

- Insight into physics of the problem
- Manufacturing specifications of the component
- Less detail without losing impact of the component on flow pattern and heat transfer

Diffusers

Here is an example of diffuser characterization. This diffuser has many small air gaps and thin metal vanes that would require a small element size to capture flow accurately. The diffuser has small details w.r.t room it is supplying and explicitly modelling these details would add to CFD meshing and solving complexity. Hence such a component can be characterized as shown in the image below. A single diffuser is modelled explicitly, and velocity components are obtained. These velocity components become inputs to the simplified model where in inputs are velocity components and geometry details are not explicitly modelled.

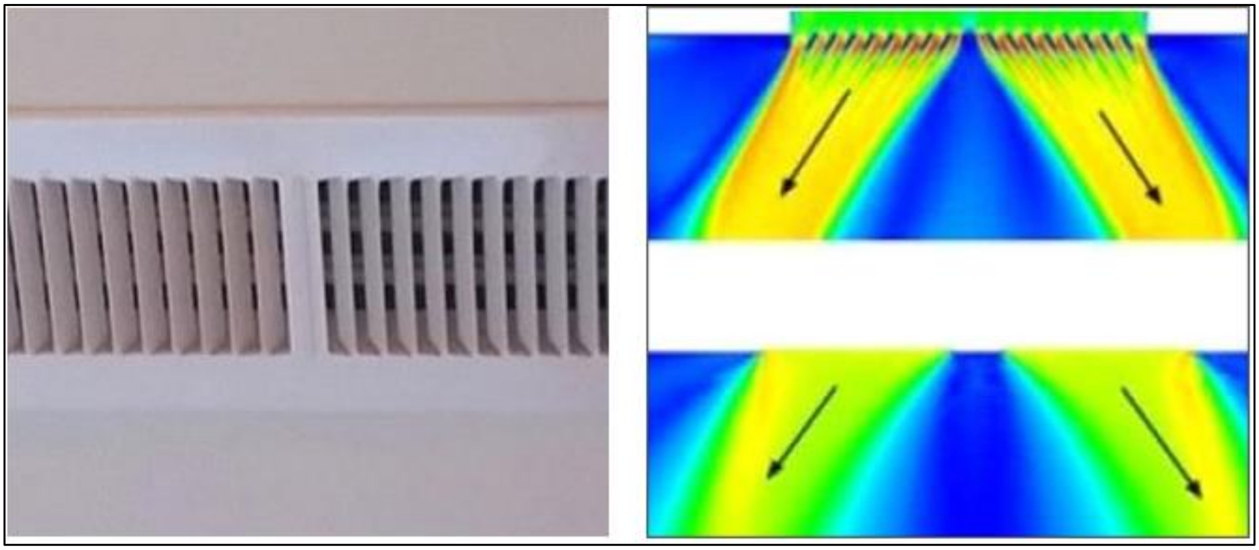
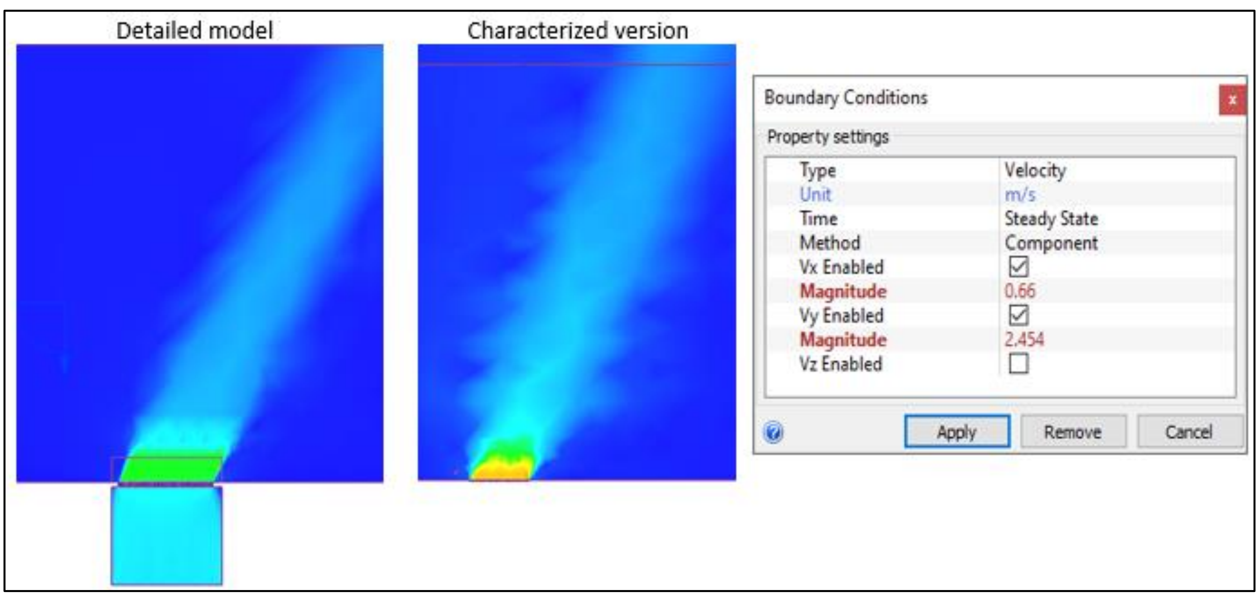


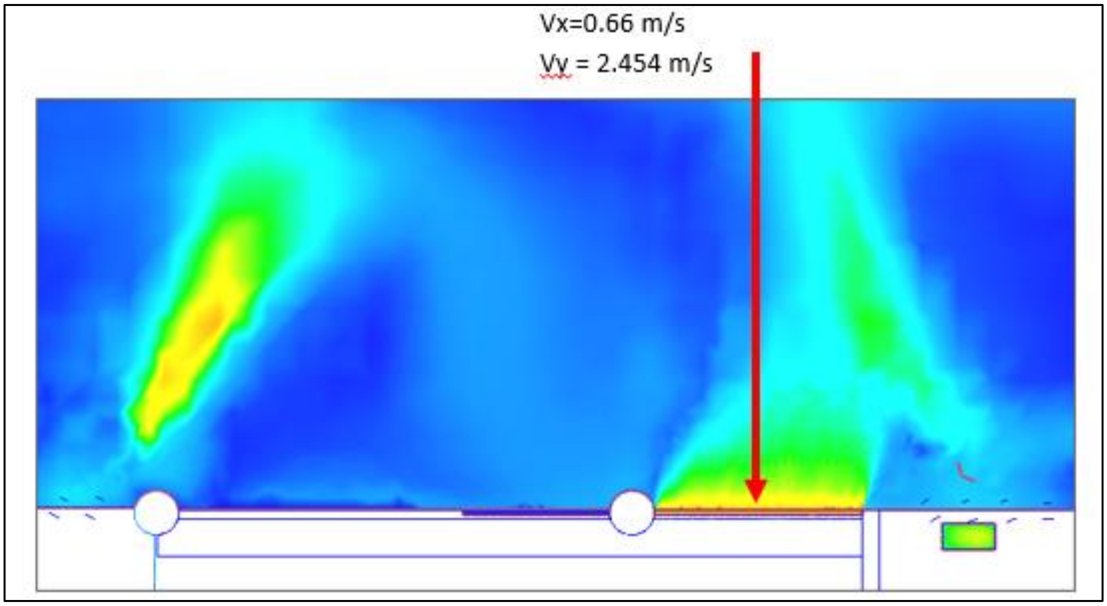
Image credit: Microgenesis

Diffusers

Inlet velocity converted to components based on diffuser angle to get required flow direction and CFM.



Velocity results comparison - Detailed Vs Characterized version



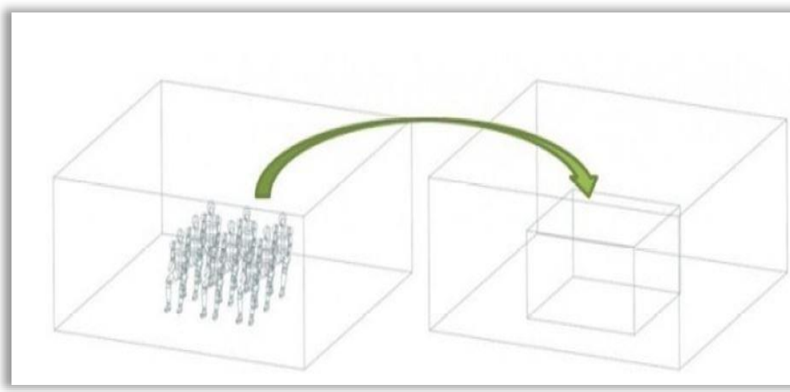
Use of characterization of diffusers

People

Airport departure and arrival halls are circulation areas where people keep moving about for check-in and baggage reclaim. Modelling occupants with all possible details computationally may not be practical. Hence human occupancy is modelled as:

- Air volume with height of 1.8m from floor
- Air volume is assigned heat generated by occupants

Image credit: Microgenesis



People

AUTODESK UNIVERSITY

Appropriate Physics

Mixed Convection

A best practice that could be very important in terms of appropriate physics is use of mixed convection. Air-conditioning in PTB is by mechanical ventilation wherein air *movement* is driven by fans to provide *air changes* within the occupied space, i.e., replacing stale air with fresh air as well as removal of heat (forced convection-fixed air properties). This results in human comfort and good indoor air quality. However, there may be some pockets with free or buoyancy driven flow (natural ventilation). In CFD modeling it may be necessary to include effects of buoyancy (variable air properties) to properly capture airflow and heat removal. In this project, mixed convection approach is used to account for buoyancy in check-in hall and baggage reclaim hall.

- Initially forced convection with “fixed” properties is simulated
- Local temperature gradients can lead to appreciable buoyancy effects
- Use “variable” air properties with gravity enabled to account for buoyancy

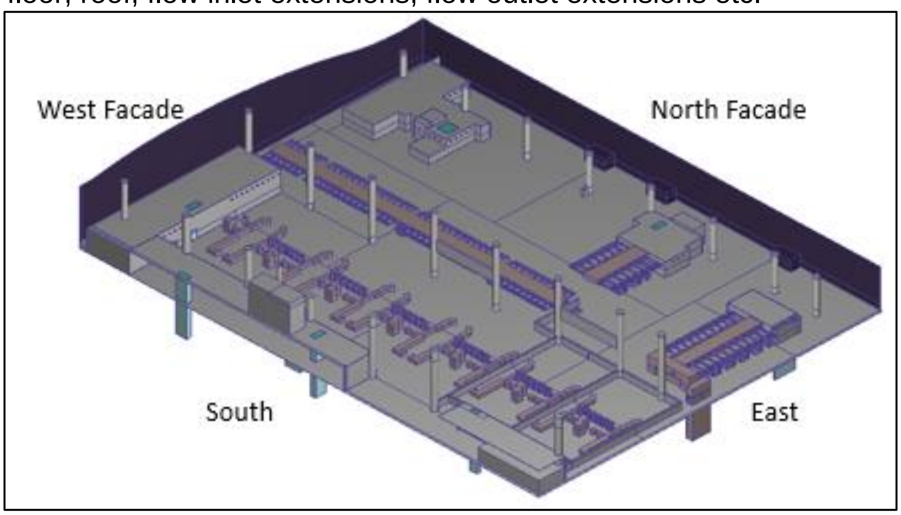
https://help.autodesk.com/view/SCDSE/2014/ENU/?guid=GUID-7BBB1E45-4469-4F90-8ED7-7756B04CEA80

Simulation strategies and techniques

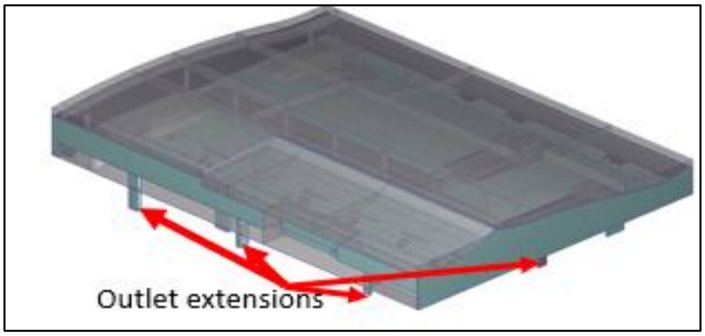
Modelling strategies or workflows we followed are basically creating a CAD model and preparing CFD model with component characterization, materials assignment, boundary conditions specification, meshing and solver settings.

CAD model

From Revit, CAD model suitable for CFD simulation is prepared. CAD model for check-in hall is shown in image below. Input details for CFD model include building geometry from CAD data with major architectural layout, interior details such as: check-in counters, service counters, rooms, walls, floor, roof, flow inlet extensions, flow outlet extensions etc.

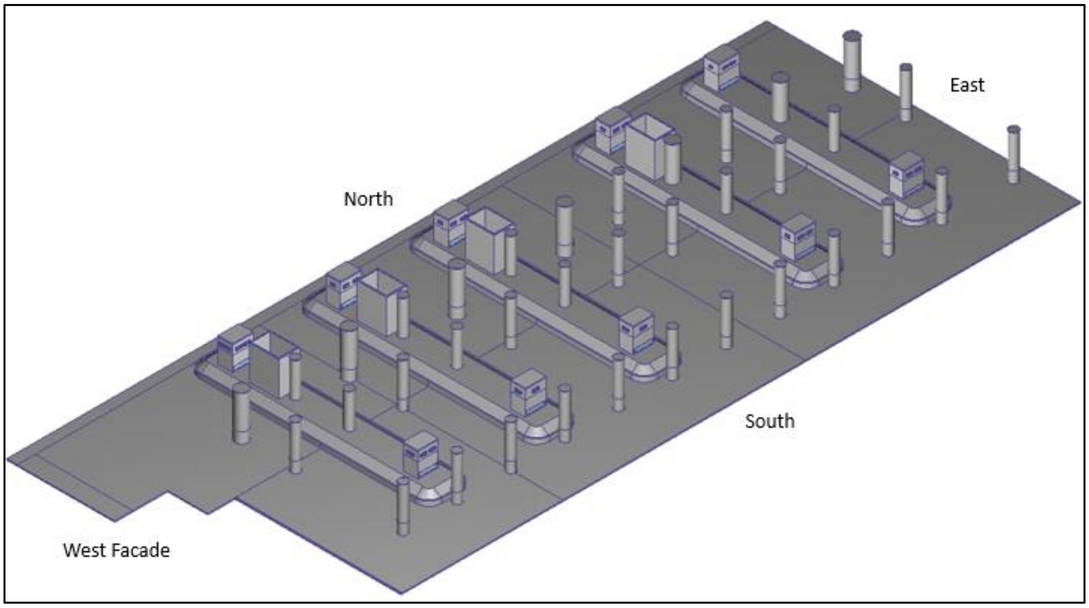


Check-in hall



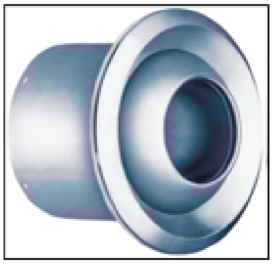
Inlet and outlet extensions created in CAD model

Simplified CAD model for baggage reclaim hall is shown in image below. We have included relevant details for flow and heat transfer such as baggage conveyors, binnacles, columns, and other building details.



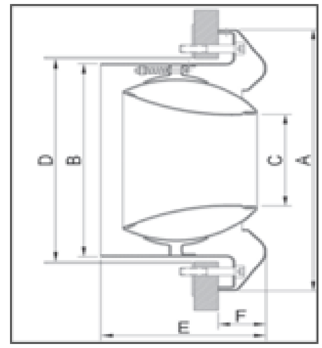
Baggage reclaim hall

Two types of supply diffusers are used in the check-in hall, namely, nozzle jet diffusers and slot diffusers.



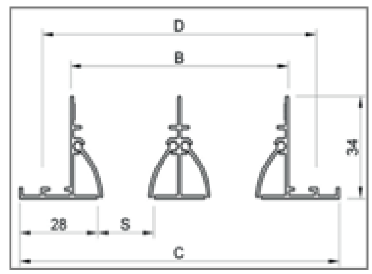
Jet Diffusers

DIMENSION DETAIL						
NOM	A	B	C	D	E	F
200	268	198	94	208	138	40
280	345	275	132	283	140	40
320	395	318	160	328	190	42
400	468	397	224	407	224	46



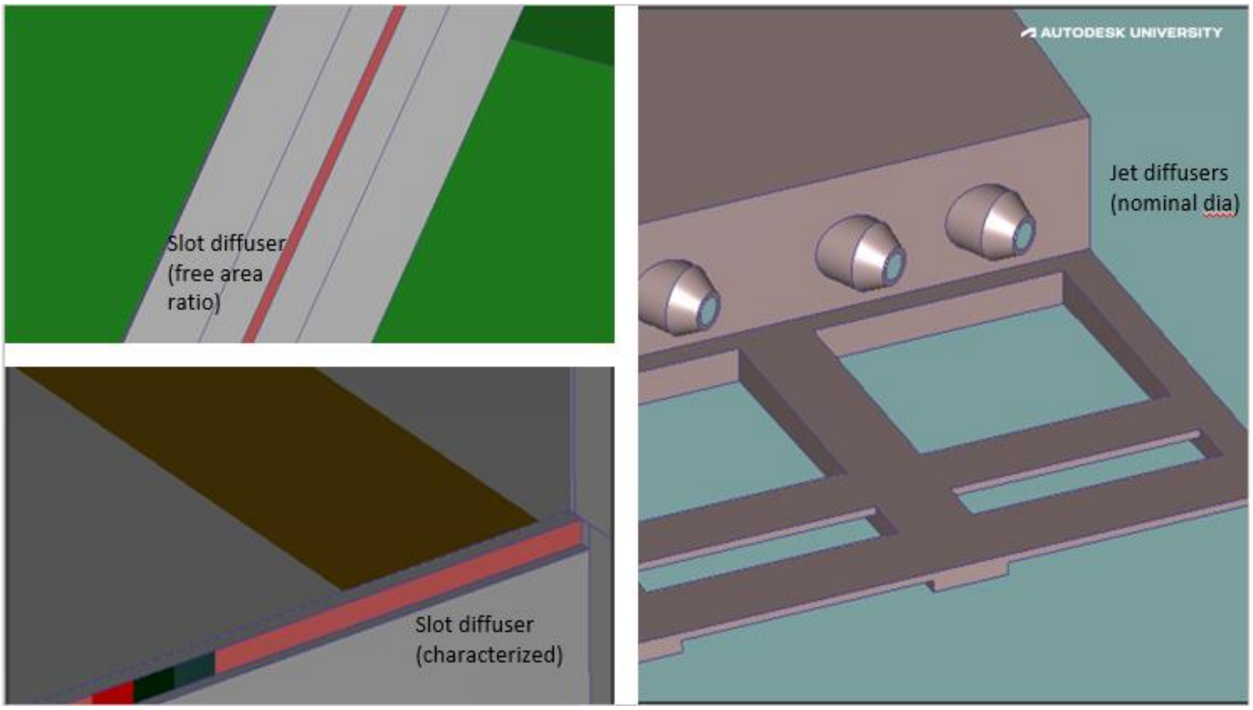
Slot Diffusers

No. of Slots	B	C	D
1	40	76	56
2	48	115	94
3	117	153	133
4	155	192	171
5	194	230	210
6	232	269	248
7	271	307	287
8	309	346	325



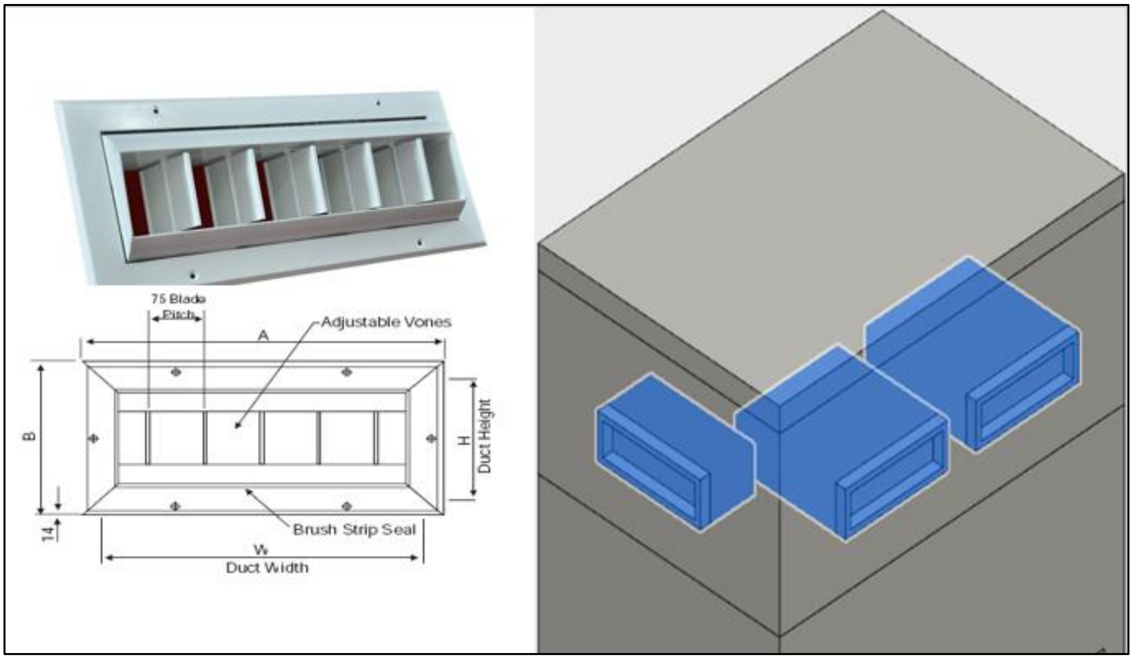
Diffuser specifications

Nozzle jet diffusers are modelled as per actual geometry for the supply of conditioned air whereas slot diffuser is modelled as characterised geometry. When the flow is in normal direction in slot diffuser, CFM with free area is used. When the flow is angled, velocity components are applied on the characterized geometry to capture the flow.



Simplified jet diffusers: Check-in hall

Drum louvers are modelled as characterized geometry with free area ratio if flow is normal to the drum louver supply surface. When the flow is angled, velocity components are applied to the characterized geometry to capture the flow.



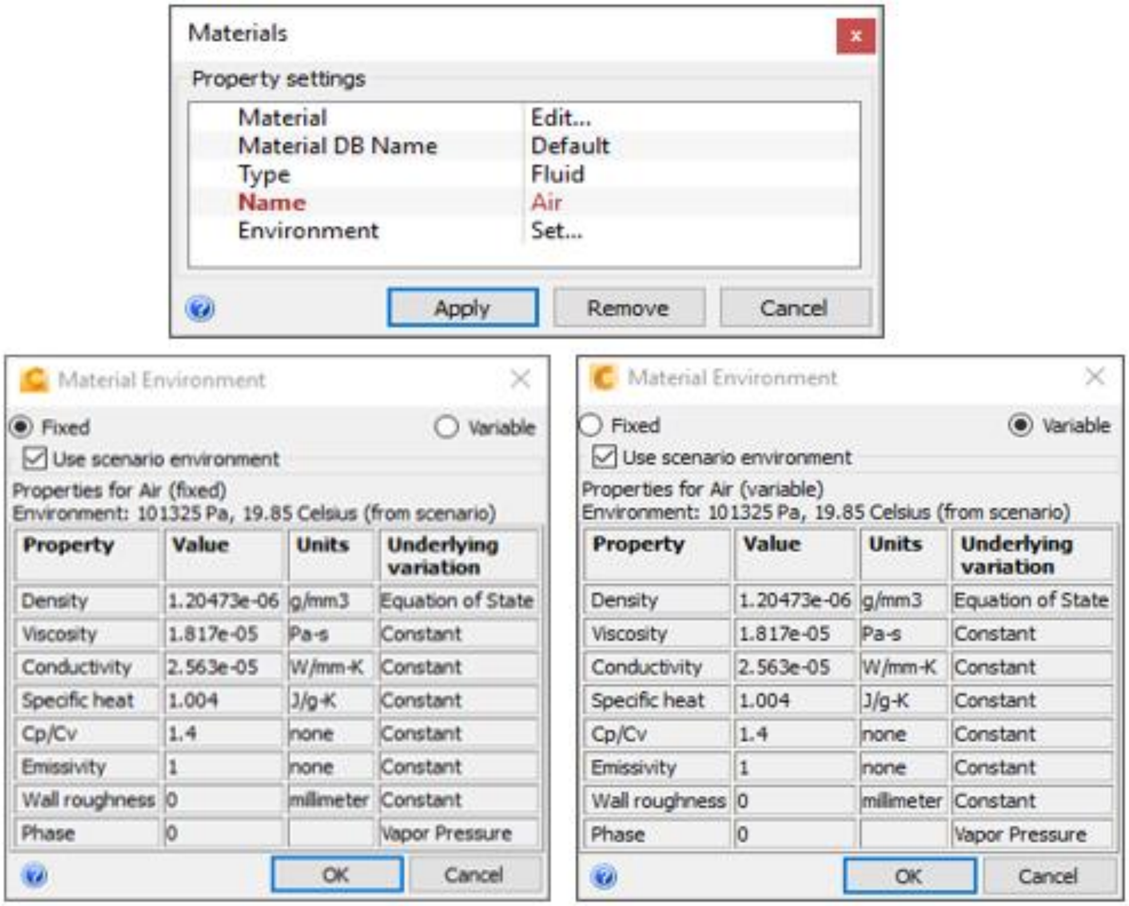
Simplified drum louvers: Baggage reclaim

Materials assignment

Materials define the properties of parts which impact the physics of simulation. Simulation CFD provides a comprehensive library of materials along with the ability to define custom materials as needed.

For AEC applications, “air” is by far the most common fluid material. For check-in hall and baggage reclaim hall CFD simulation, fluid domain is assigned “air” as material with fixed properties (i.e., density does not vary with temperature).

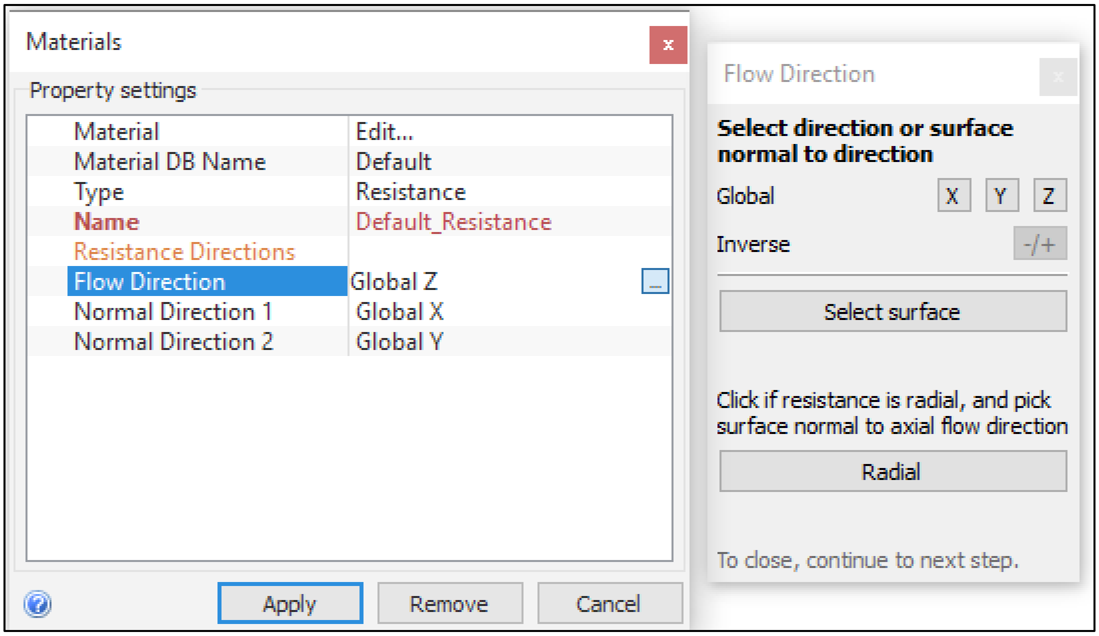
Based on local hot spots observed during CFD simulation, air properties are assigned as “variable” (i.e., density varies with temperature).



Fluid volume assigned “Air” properties as fixed and variable for mixed convection heat transfer

Outlet grills are assigned default resistance material to give resistance to the flow with a free area ratio of 0.5. i.e., 50% open area to flow.

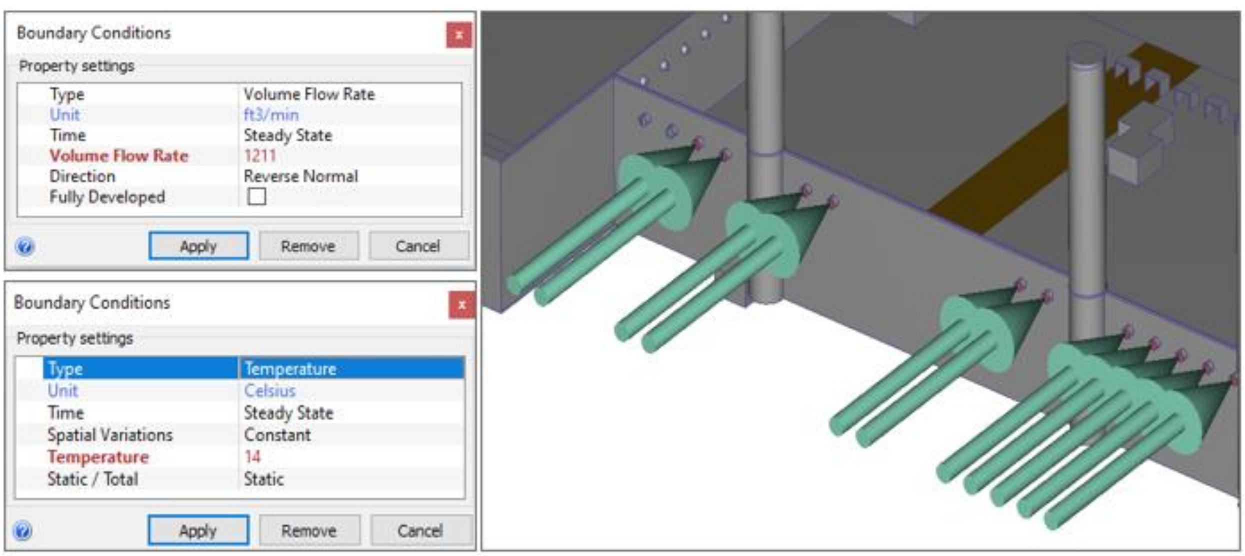
Autodesk University



Outlet grill properties

Boundary Conditions (BCs)

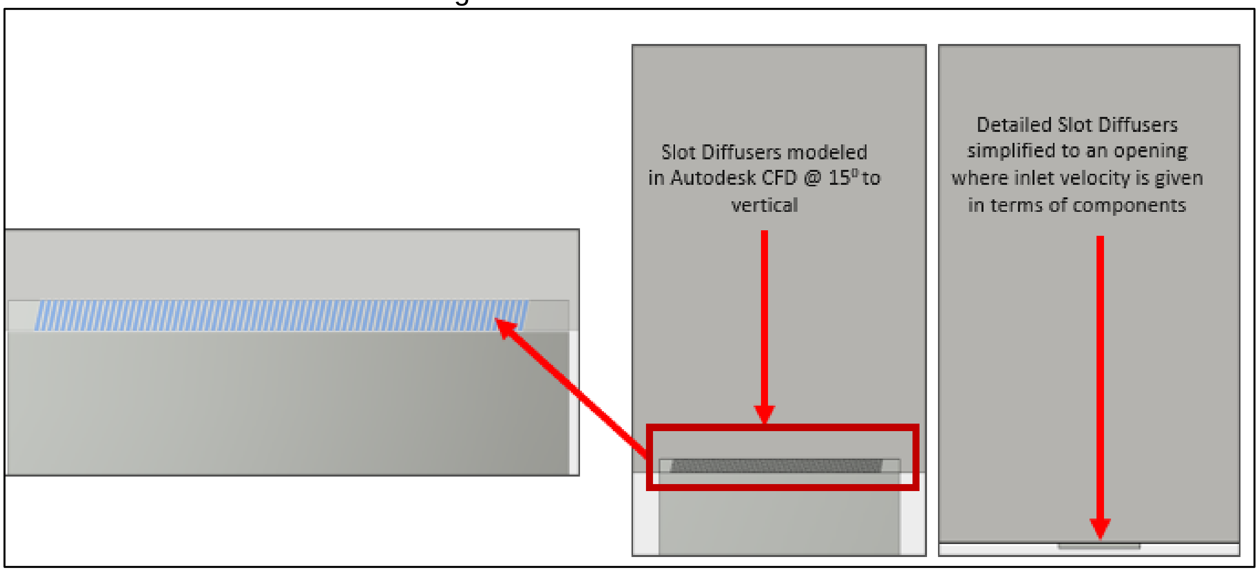
Air is mechanically moved in and out of the building space, so inlets are assigned *volume flow rate* on each supply/inlet diffuser. Arrow direction indicates the flow in the right direction from supply/inlet diffusers i.e., direction is into the occupied space.



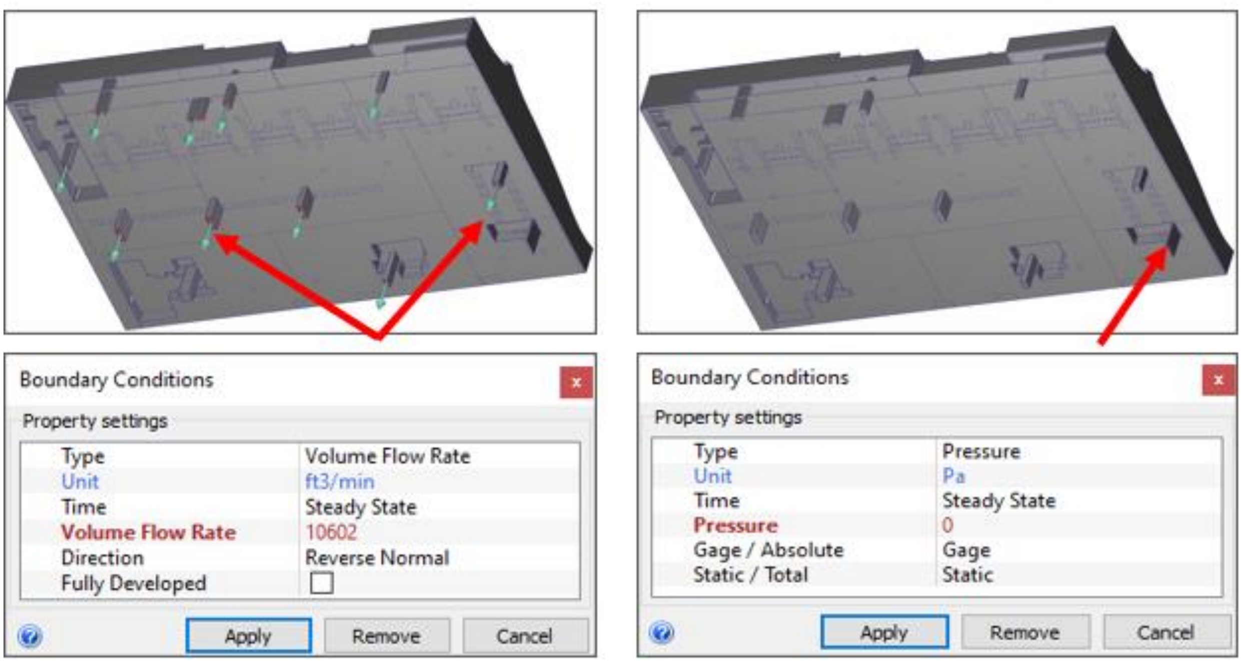
Inlet BCs- Jet diffusers for baseline design

Slot diffusers with angled flow are applied with velocity components after characterization and testing with detailed geometry. Hand calculations can be used to determine the component velocities based on the following diffuser specifications:

- Angle of flow leaving diffuser
- Incoming flow rate
- Area that the flow will be coming from



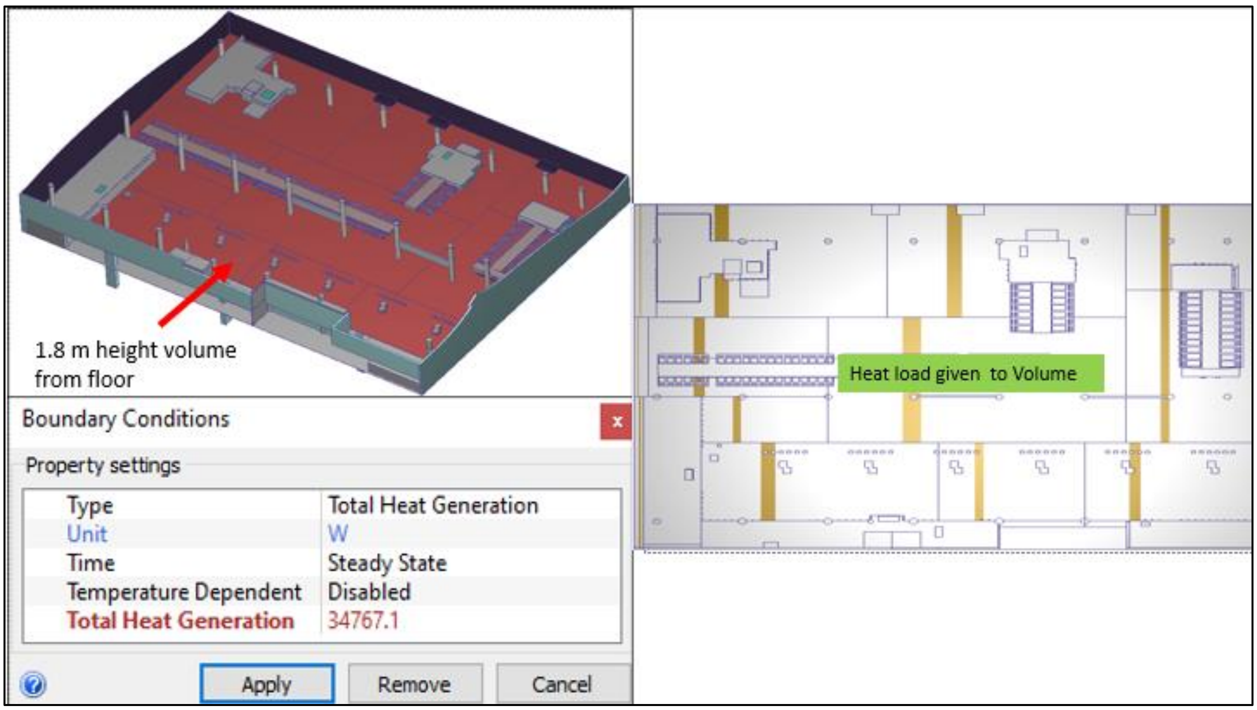
Inlet BCs - Characterization- for optimized design



Outlet BCs - Pressure and Temperature

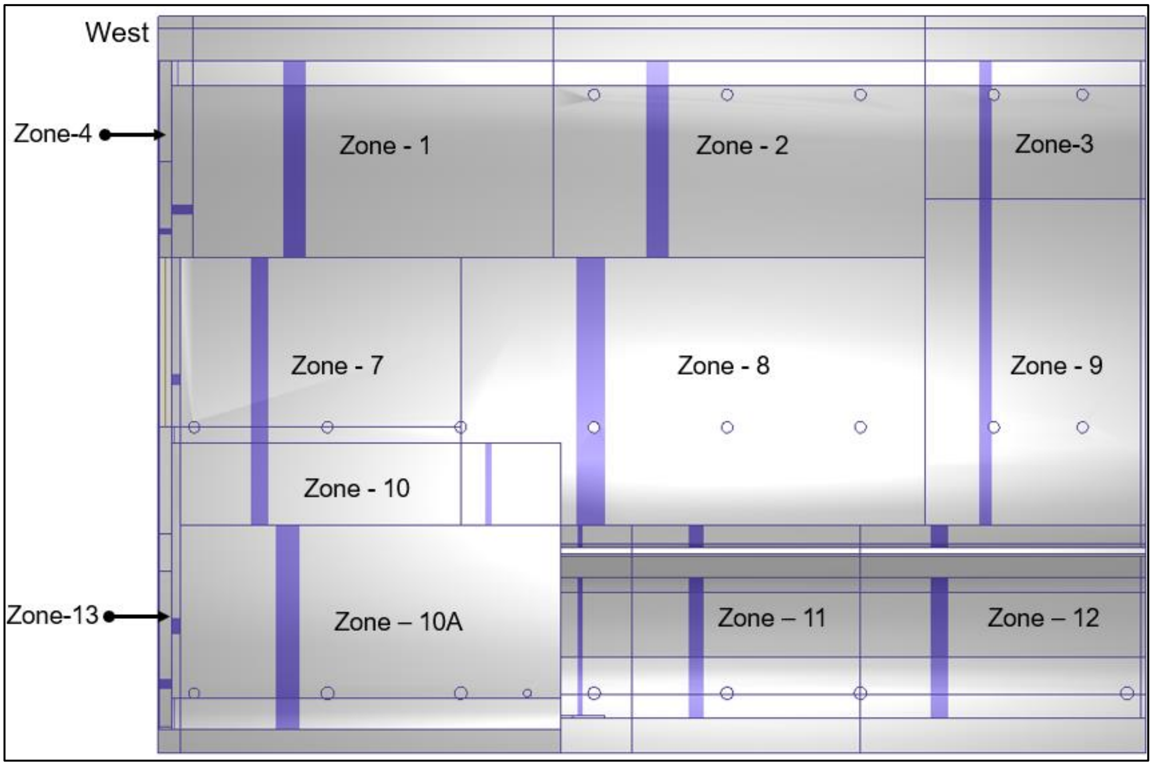
All outlets are applied with volume flow rate boundary condition except one outlet where Pressure BC is applied. This is required to get a numerical solution to flow.

Thermal simulation is driven by thermal boundary conditions and deals with internal heat gains from people, equipment, and lighting. External heat gains include solar and transmission loads. These are taken care of by defining total heat generation boundary condition for occupant volume.
People load, equipment load and solar load are applied over a volume of 1.8 m height from floor. Transmission load is applied on the façade surface in terms of U values and design outdoor ambient temperature.



Heat loads - Solar, Equipment, and Occupancy

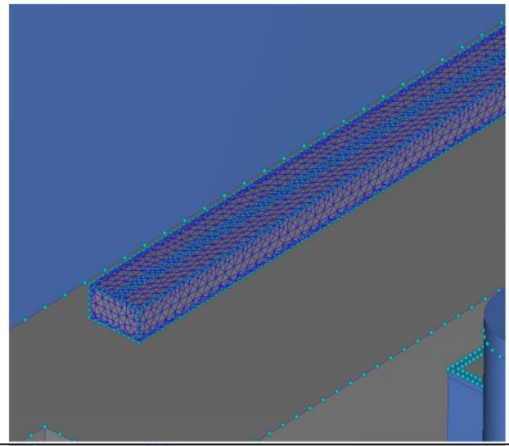
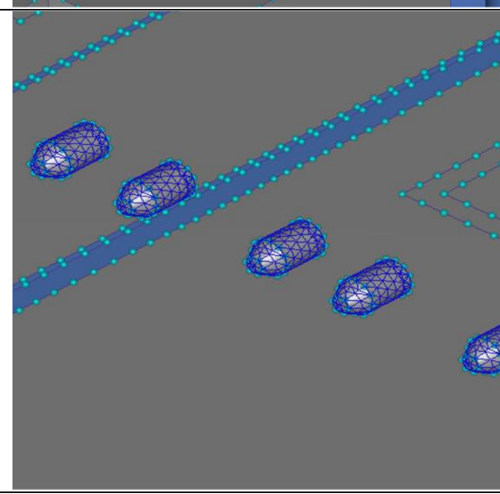
Lighting load is applied on the bottom surface of the ceiling.



Heat loads – Lighting load

Meshing

Automatic mesh sizing is used to define the mesh distribution in the CFD model domain. CFD simulation uses FEM (Finite Element Method) to calculate the fluid flow and thermal results. The FE (Finite Element) mesh is the backbone of CFD simulation/calculation and has a direct impact on solution accuracy. In meshing task, the model geometry is divided into many smaller regions called elements, where each corner of element is a node at which flow, and thermal variables will be calculated. For simulation, a certain number of elements will be required to adequately capture flow and thermal characteristics. As element count increases, so does the solution time and hardware requirements. Question is how many elements would be required? Well, we use just enough number of elements for mesh independent solution. For better accuracy, first we do automatic mesh sizing for the entire domain and then we do refinement on volumes as necessary. Local mesh refinement with uniform mesh is used for mesh on supply diffusers to capture the flow through them. It is important to mesh inlet and outlets with 4-5 elements to capture flow properly. Also, jet diffusers, slot diffusers as well as outer grills have uniform mesh with 4-5 elements. Fine mesh is applied in regions in front of the diffusers.

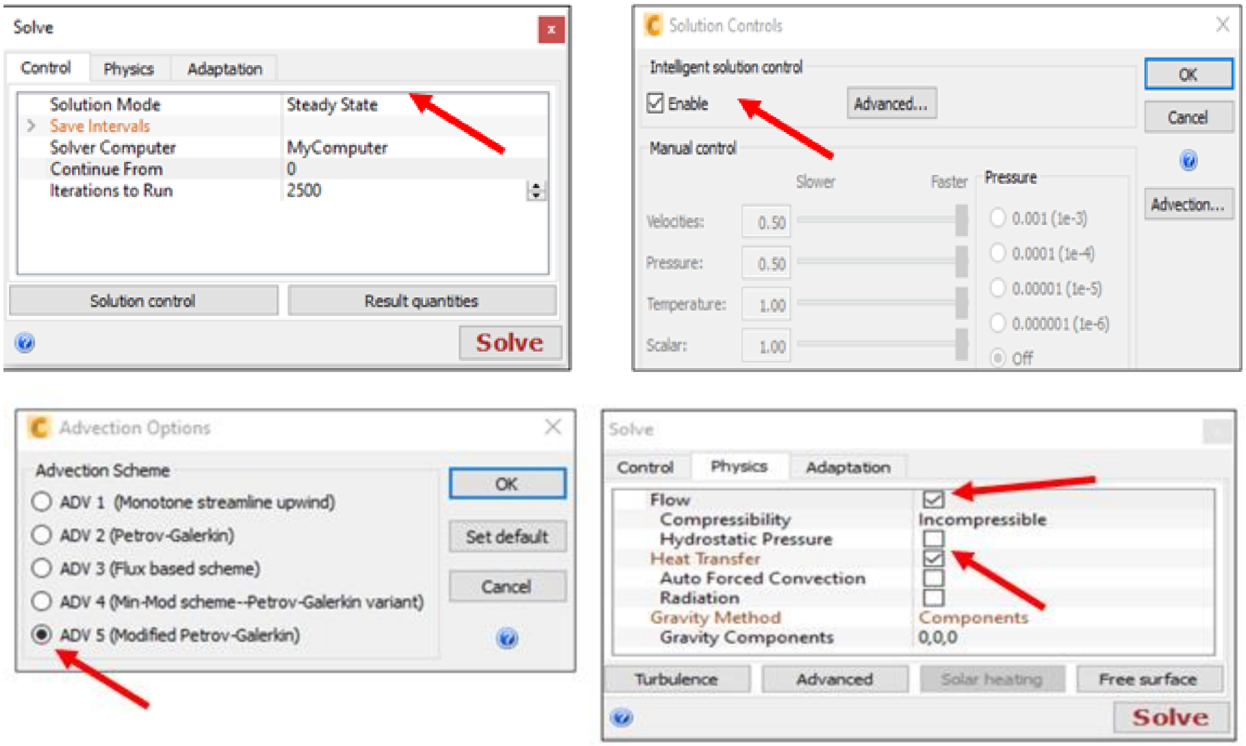
Fine mesh applied below slot diffusers to capture gradients	
Uniform mesh to capture flow from Jet diffusers	

Local mesh refinement

Solver settings

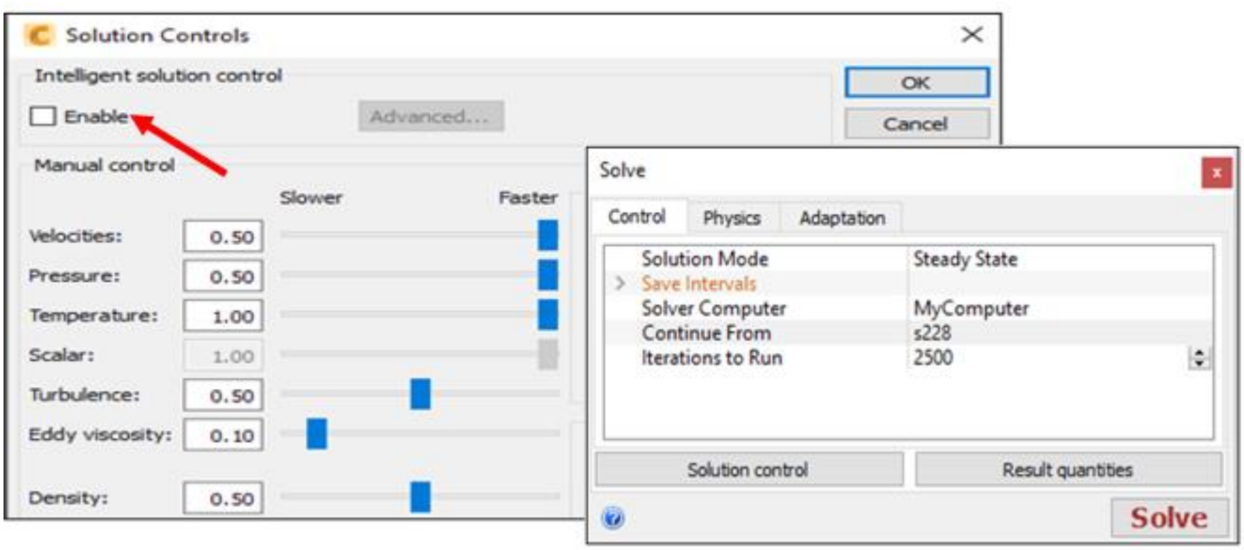
The solve dialog in the simulation task is the command centre for the simulation. It controls the physics of simulation and what data is to be output. The solve dialog is comprised of 3 tabs…control, physics, and adaptation. Each of which have a variety of input fields and buttons. Simulation CFD has the capability of solving a wide variety of fluid flow and heat transfer applications. The first step in assigning solver settings is to determine the physics of the problem being solved. In this case, initially steady state flow with fixed air properties and ISC switched on is solved.

Heat transfer needs to be considered in this analysis, so heat transfer is enabled in the solution settings with flow as incompressible. Advection 5 scheme is used as it is the recommended scheme for most flow analysis.



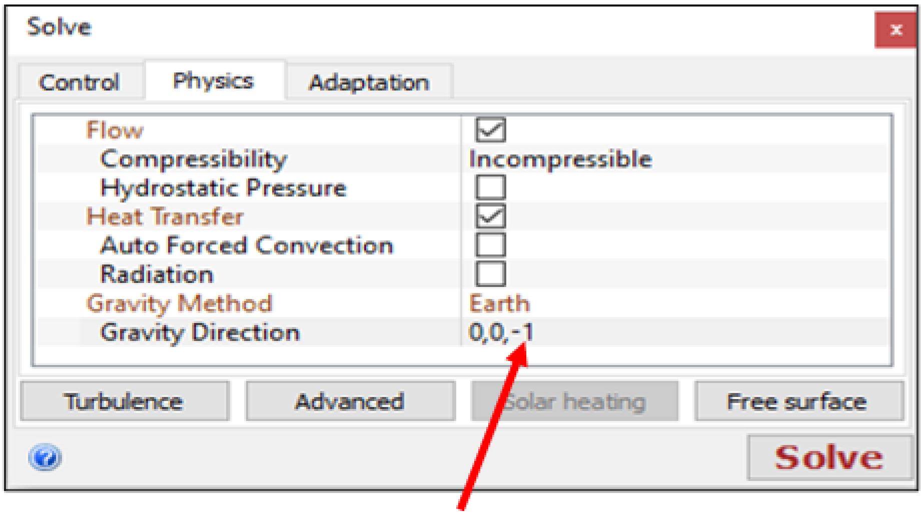
Solver settings for flow analysis (Intelligent solution control (ISC)-On, Advection Scheme 5)

After solving for certain number of iterations, temperature field is checked and if local hot spots are present, then air properties are changed to “variable” with ISC switched off.



Solver settings for flow analysis (Intelligent solution control-Off)

Buoyancy effects due to gravity need be considered. So, gravity is enabled, and heat transfer is by mixed convection.



Gravity enabled to account for buoyancy effects

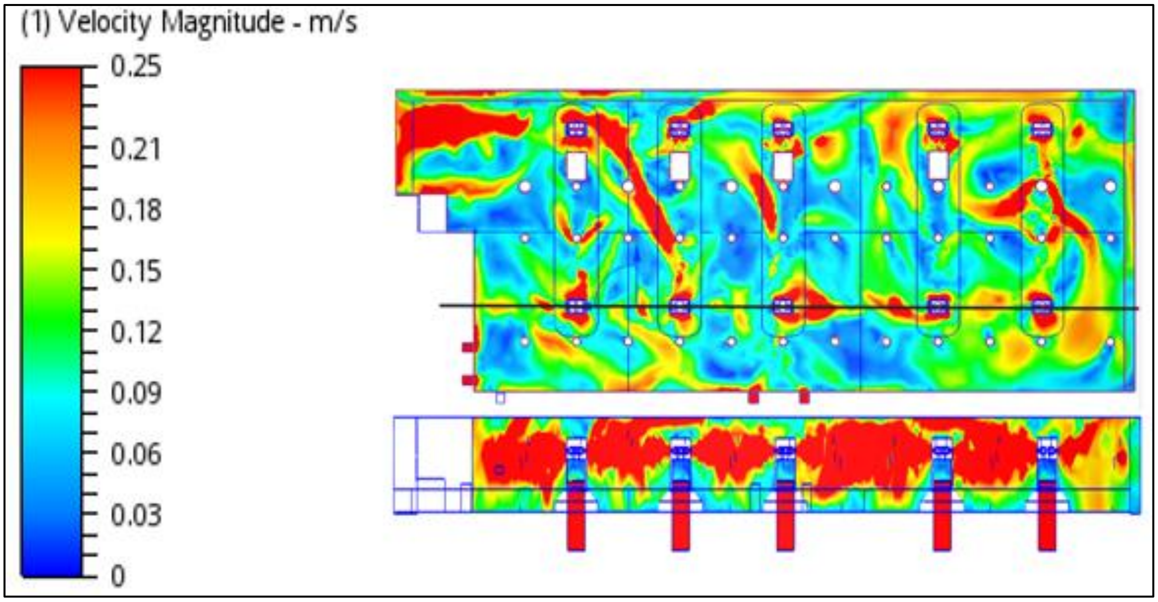
Mixed convection

Key results for design performance

After simulation is set up, run, and converged, we can look at results using "Results Visualization" tools in Autodesk CFD simulation. We can visualize performance characteristics such as temperature, velocity that are difficult to capture in the real world at all locations.
Air flow velocity and temperature values are the key results from CFD analysis to evaluate thermal comfort.

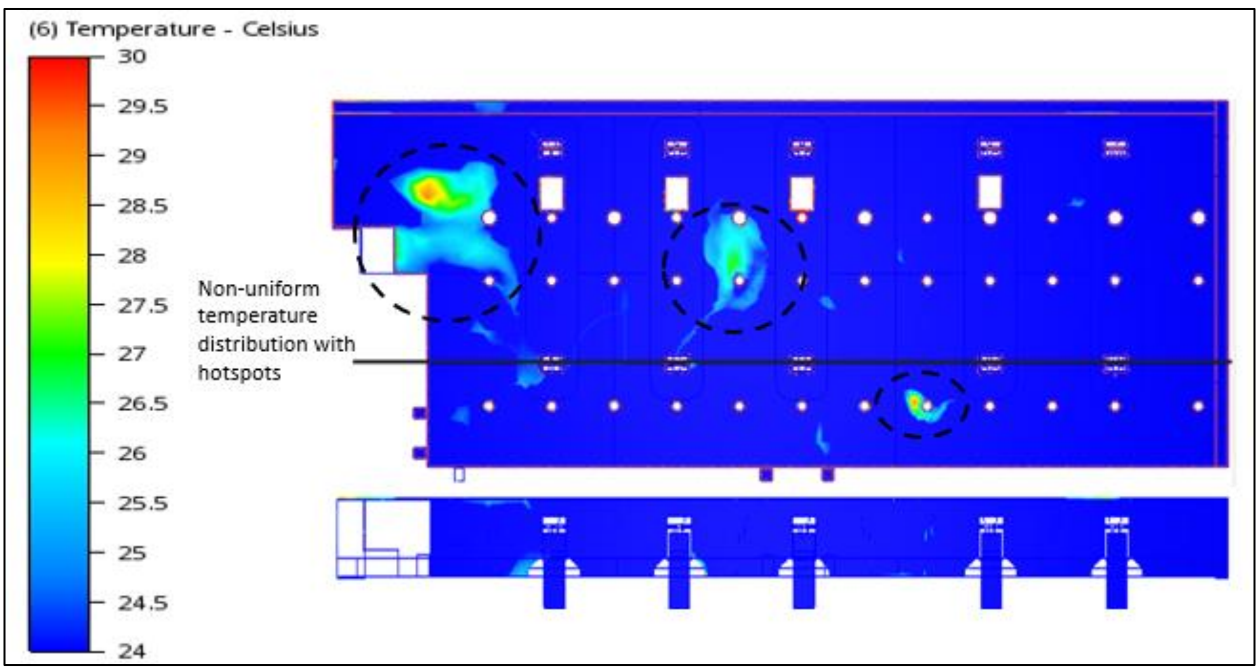
Baggage reclaim: Baseline design

CFD results for baseline design showed hot spots near west façade and at few locations in between baggage reclaim area. This resulted from hot-air recirculation and temperature exceeding the design temperature limit of 24 +/- 1^{0} C at occupied space which is unacceptable.



Velocity distribution

Results Plane in this image provides velocity distribution in the occupied space for baggage reclaim hall for baseline design at **1.8m from floor**. This helps us to look at local values and variation from point to point.

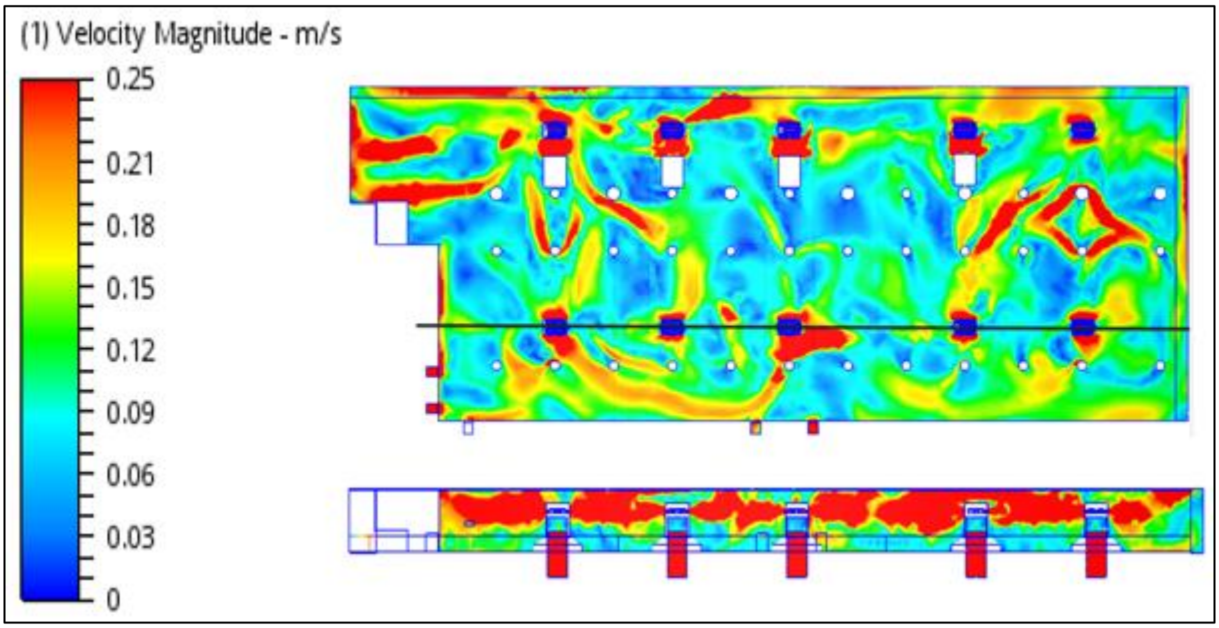


Temperature distribution

Results Plane in above image provides temperature distribution in the occupied space for baggage reclaim hall for baseline design at **1.8m from floor**. This helps us to evaluate local variation of thermal comfort condition in terms of temperature. Here we notice hot spots with temperature above design requirement which is not acceptable.

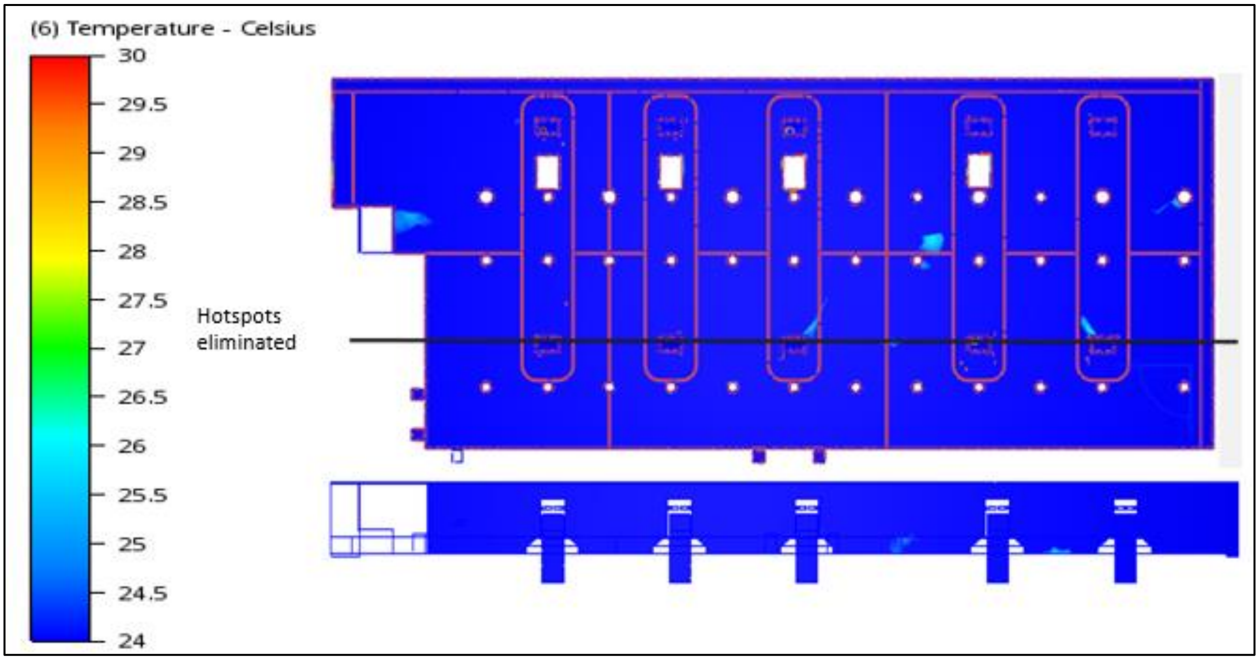
Baggage reclaim: Optimized design

CFD results for optimized design showed temperatures at occupied space within design limit i.e., 24 +/- 1^0 C due to diffuser angle directed towards the hot spots. This resulted from better movement of conditioned air at occupied space at **1.8m from floor**.



Velocity distribution

Results Plane in this image provides velocity distribution in the occupied space for optimised design for baggage reclaim hall at **1.8m from floor**.

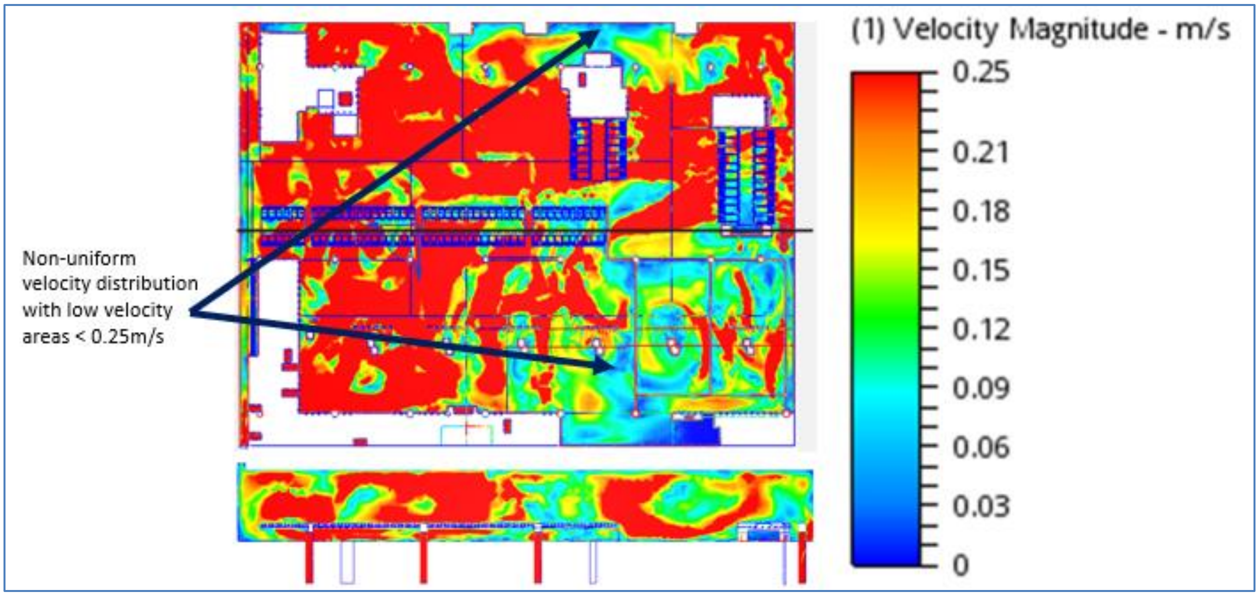


Temperature distribution

Results Plane in this image provides temperature distribution in the occupied space for optimised design for baggage reclaim hall at **1.8m from floor**. Hot spots are eliminated, and occupied space meets the indoor design temperature requirement.

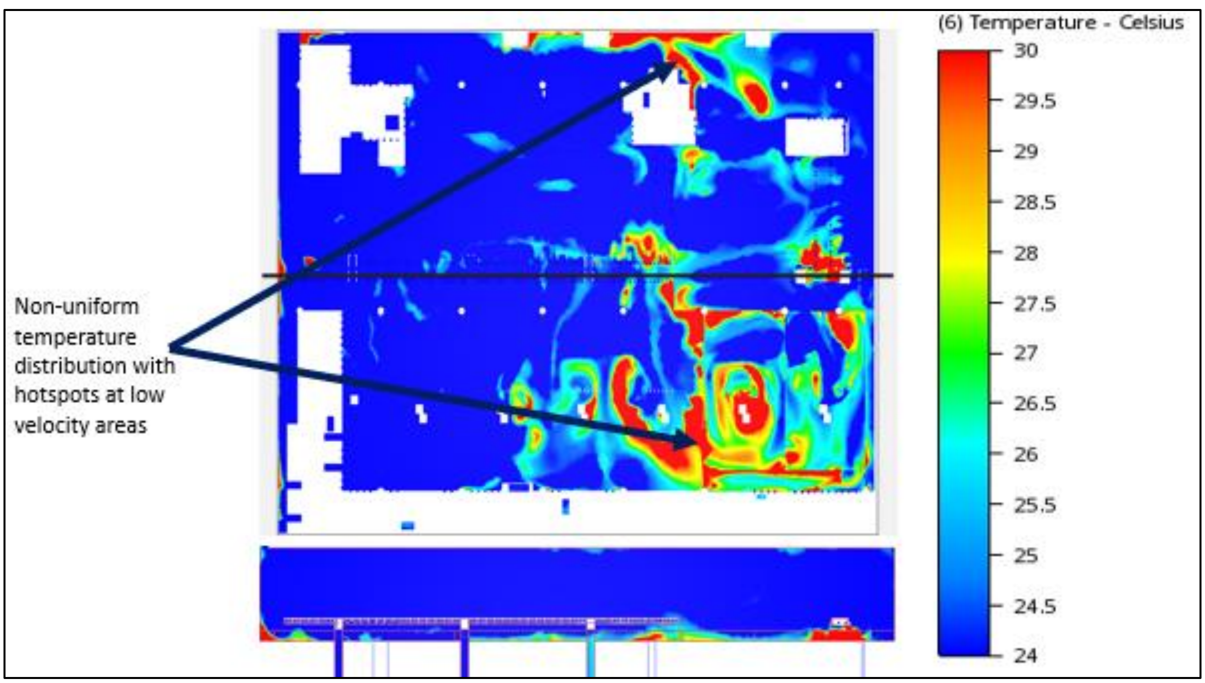
Check-in hall: Baseline design

CFD results for baseline design showed hot spots near screening area/south wall of check-In hall and few locations near north façade. The temperatures in these regions have exceeded the design temperature limit of 24 +/- 1^0 C.



Velocity distribution

Results Plane in this image provides velocity distribution in the occupied space for check-in hall for baseline design at **1.8m from floor**. Non-uniform velocity distribution with low velocity areas (< 0.25m/s) is observed.

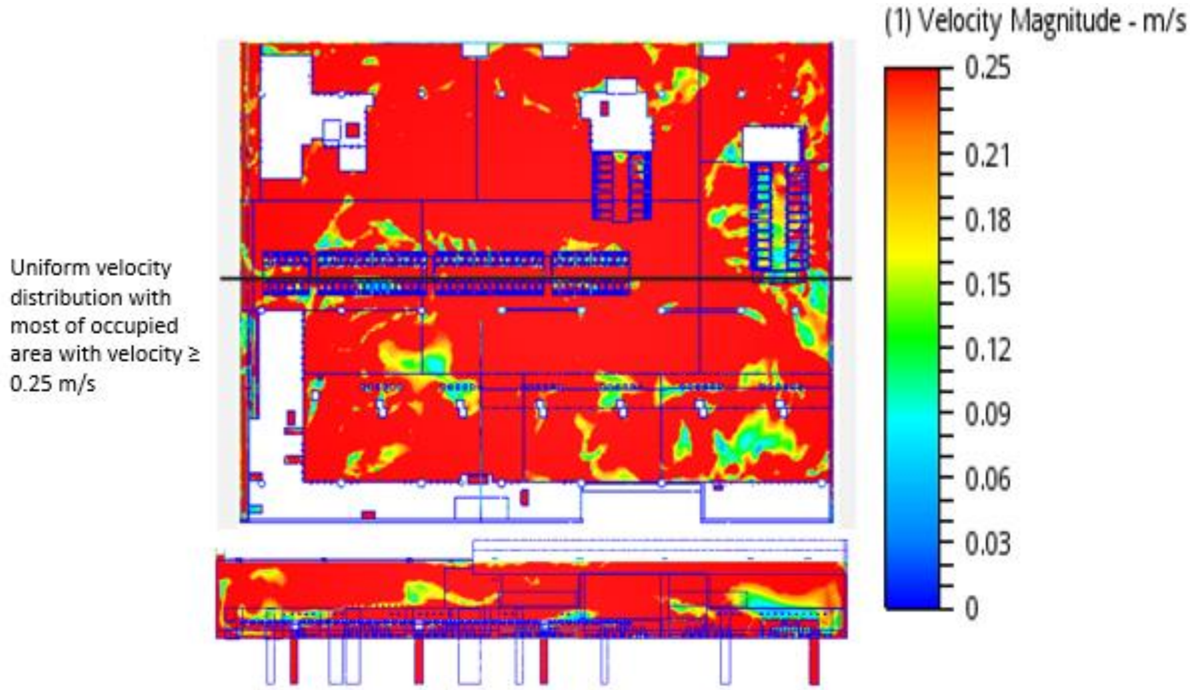


Temperature distribution

Results Plane in this image provides temperature distribution in the occupied space for check-in hall for baseline design at **1.8m from floor**. Non-uniform temperature distribution with hotspots at low velocity areas is observed.

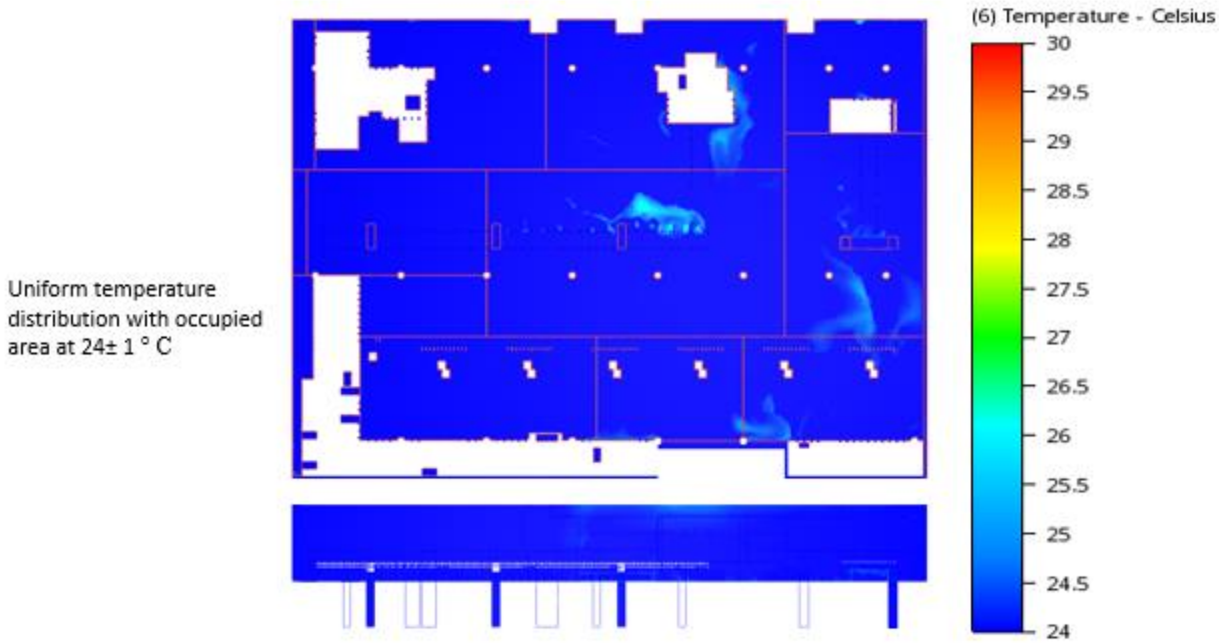
Check-in hall: Optimized design

CFD results for optimized design showed temperatures at occupied space with design limit i.e., 24 +/- 1^0C. Change in diffuser angle directed towards the hot spots as well as Inlet volumetric flow rates resulted in better air movement at occupied space at 1.8m from floor.



Velocity distribution

Results Plane in this image provides velocity distribution in the occupied space for optimized design for check-in hall at **1.8m from floor**. Uniform velocity distribution with most of occupied area with velocity ≥ 0.25 m/s is achieved.



Temperature distribution

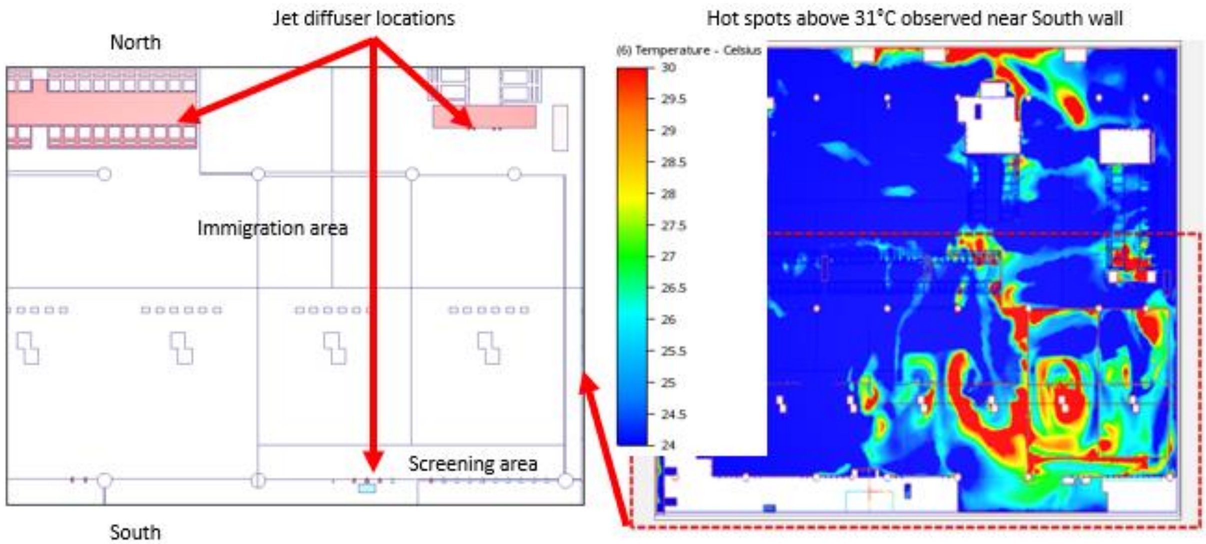
Results Plane in this image provides temperature distribution in the occupied space for optimized design for check-in hall at **1.8m from floor**. Uniform temperature distribution with occupied area at 24± 1 ° C is achieved.

Opportunities for design optimization: Check-in hall

So far, we have seen that optimized design provides required indoor design temperature. In the next section, we will go through how we arrived at the optimized design for check-in hall, though similar procedure is followed for baggage reclaim hall also.

CFD results for baseline design

CFD results for baseline design showed hot spots near screening area/south wall of check-In hall and few locations near north façade. The temperatures in these regions have exceeded the design temperature limit of 24 +/- 1^0C and are in the range of 30^0 C to 35^0 C. Here you can see that baseline design results indicate hotspots in the occupied space over 31^0 C which is clearly not acceptable.

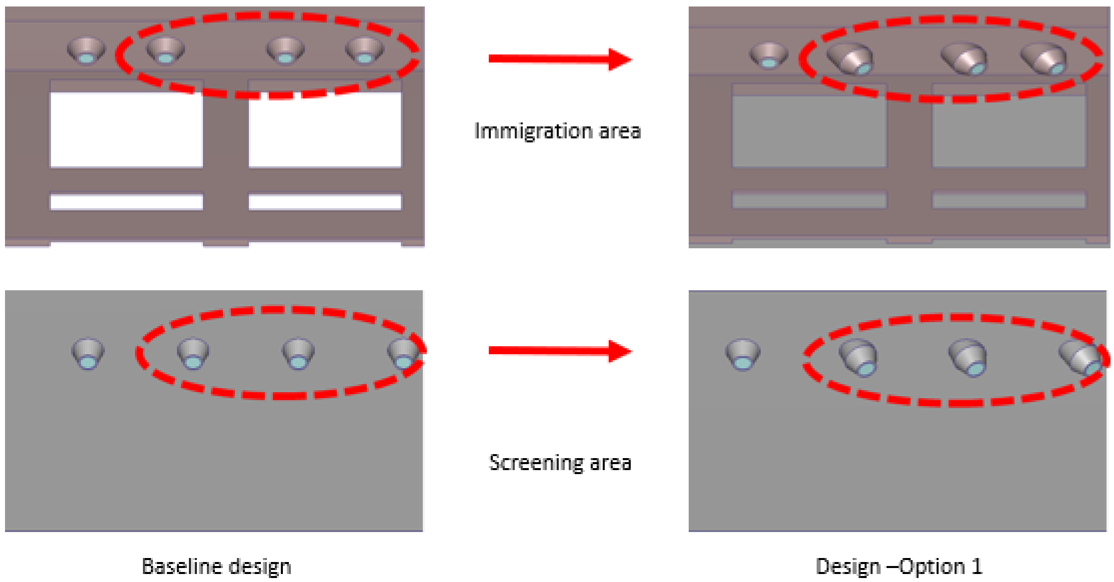


Temperature results – baseline design

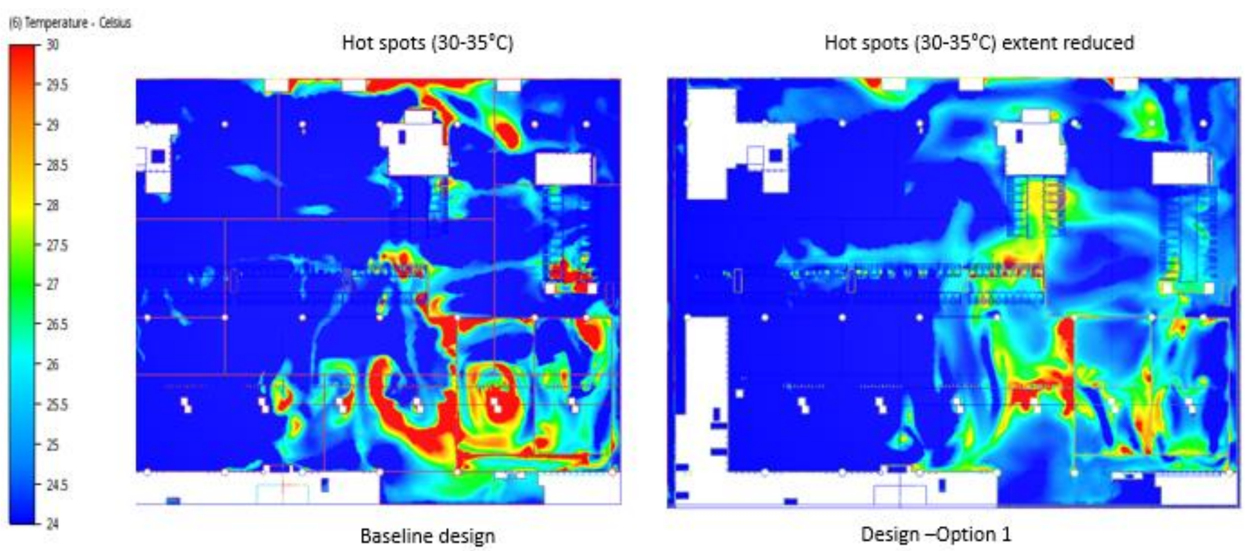
CFD results for design option 1

Baseline results are not acceptable, so design change is made w.r.t to supply flow CFM and flow angles to direct the flow towards hot spots. Let us call that as design option 1.

Diffuser flow discharge angle changed to 15 degrees to address high temperature regions. CFD results for this option showed that the extent of hot spots is reduced due to improved air flow.



Diffuser flow discharge angle changed to 15 degrees to address high temperature regions

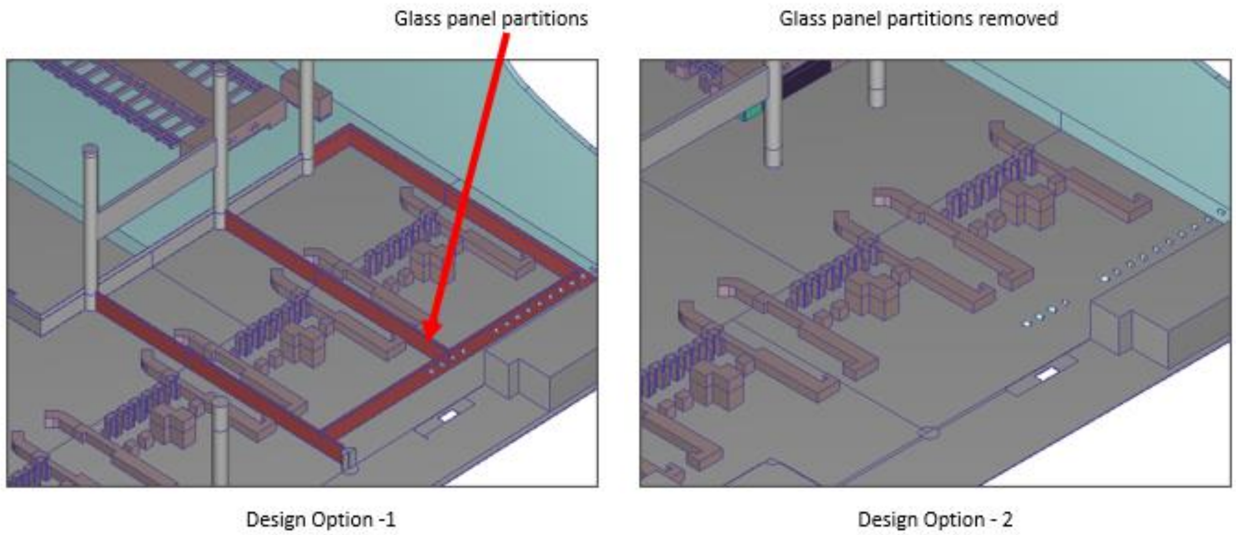


Comparison with baseline design

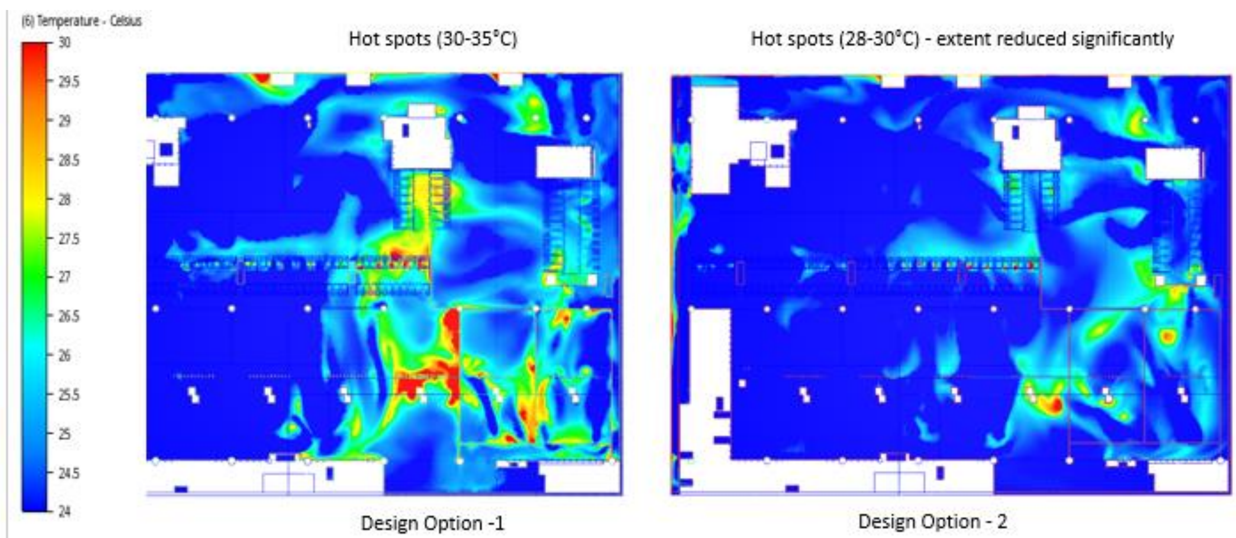
When we compare temperature results of design option1 with baseline results, we notice that extent of hot spot areas is reduced.

CFD results for design option 2

Based on improved results of design option 1, a design change is made in the architectural layout i.e., glass panels on the southside are removed for better air flow distribution.



Design change - architectural layout

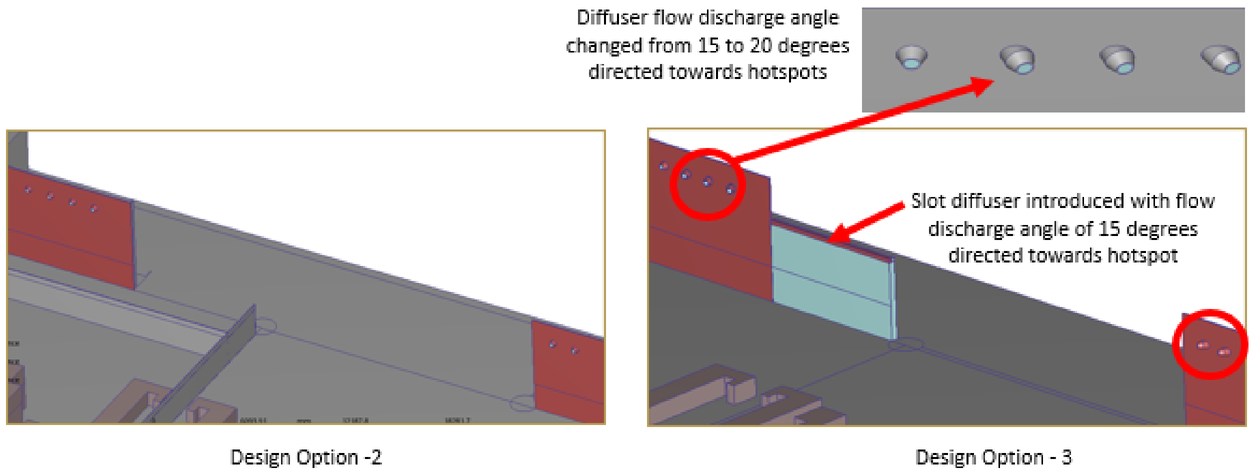


Temperature comparison with design option 1

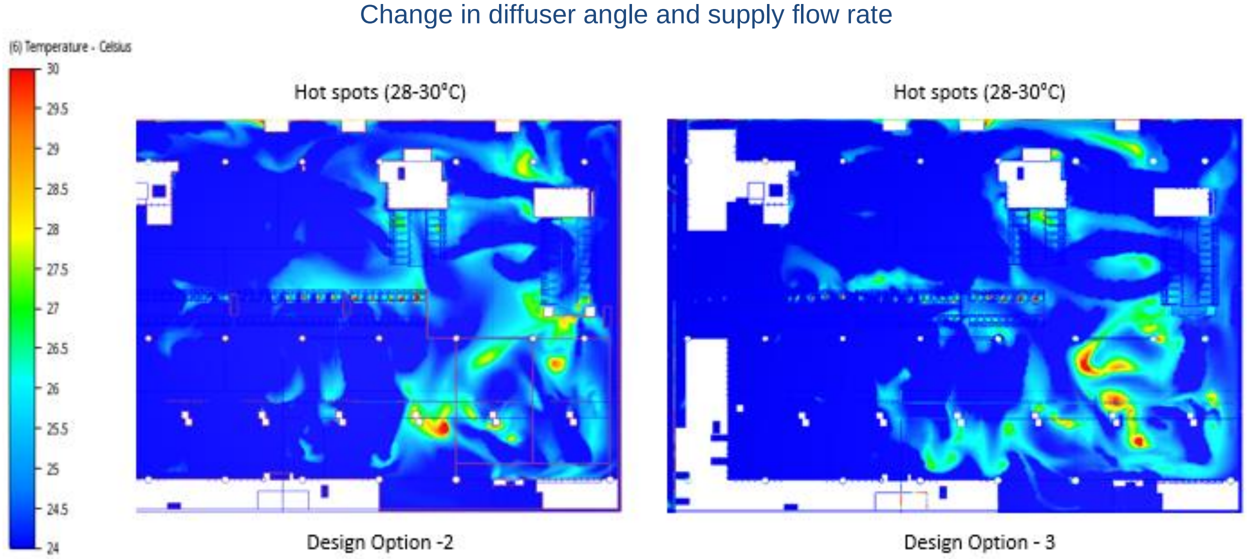
When we compare results of design option 2 with design option 1 results, we notice that extent of hot spot areas is reduced significantly. Temperature of hot spots is also reduced.

CFD results for design option 3

The jet diffuser flow discharge angle near the south wall has been changed from 15 to 20 degrees and directed towards hotspots. At slot diffuser near the south wall, CFM has been introduced with flow discharge angle of 15 degrees directed towards hotspot.



Change in diffuser angle and supply flow rate

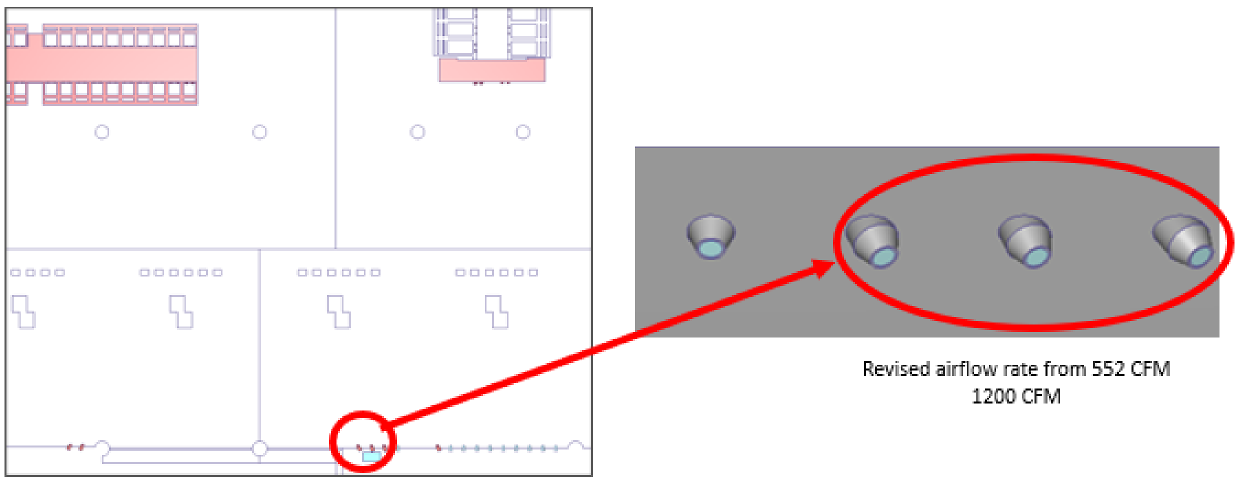


Temperature comparison with design option 2

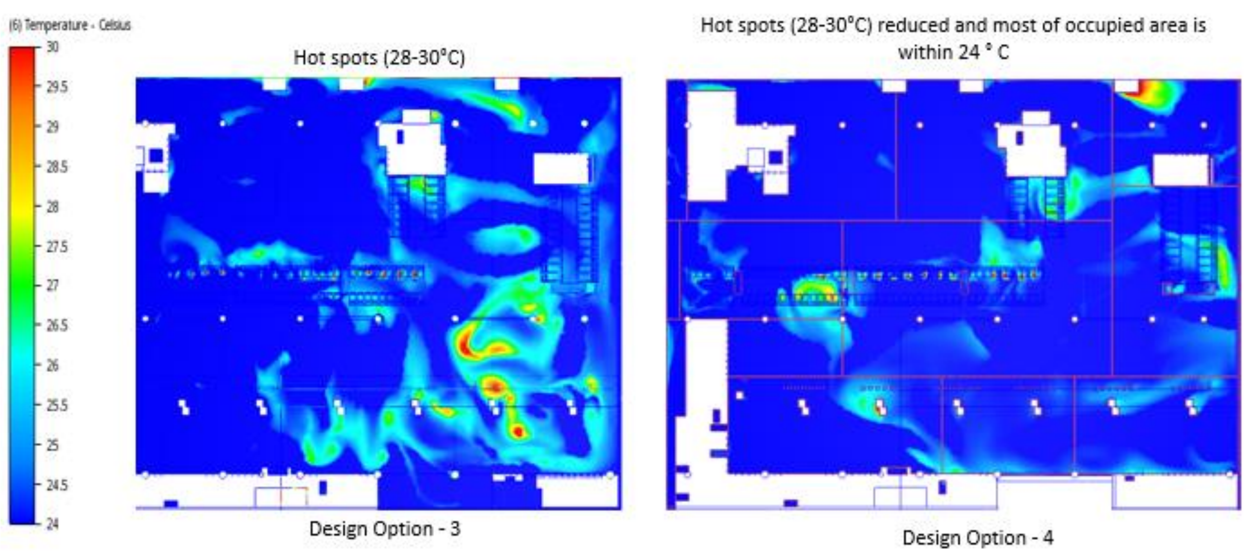
When we compare temperature results of design option 3 with design option 2 results, we notice that extent of hot spot areas is not changed much.

CFD results for design option 4

The jet diffuser flow discharge CFM at the south wall has been changed from 552 CFM to 1200 CFM and directed towards hotspots.



Change in supply flowrate

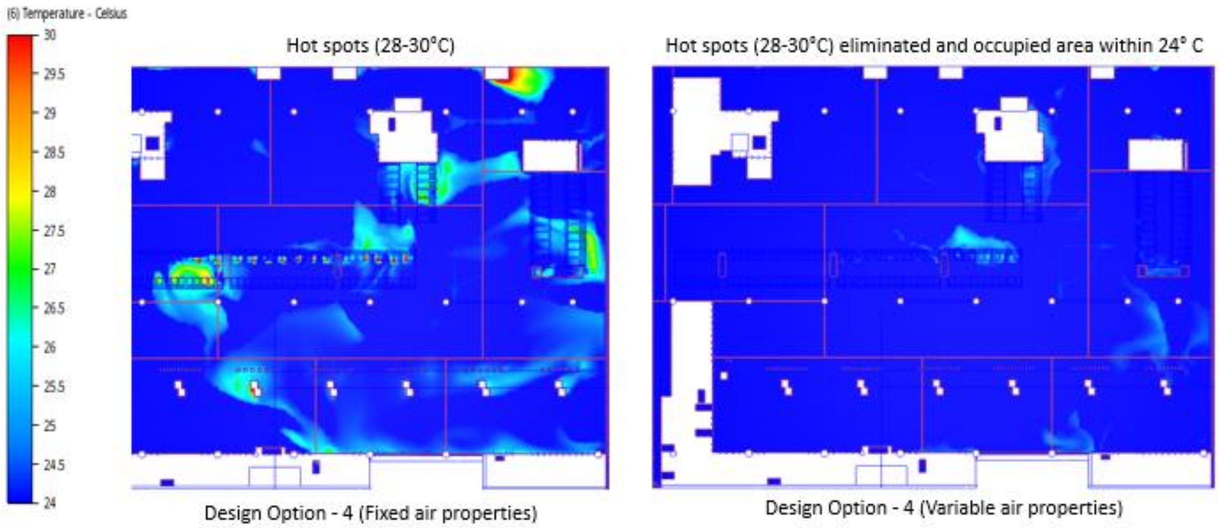


Temperature comparison with design option 3

When we compare results of design option 4 with design option 3 results, we notice that extent of hot spot areas (28-30^0 C) is reduced and most of occupied area is within 24^0 C

CFD results for design option 4 with mixed convection

When we use mixed convection by changing air properties to "variable", we notice that hot spots (28-30^0 C) are eliminated and occupied area is within 24^0 C, since the buoyancy effects have been taken care of.



Effect of mixed convection

Better HVAC designs using Autodesk CFD

In conclusion, we have seen that using Autodesk CFD, we are able to evaluate a HVAC design and optimize the design to get better results, meeting performance requirements for airport expansion project.

The design of check-in hall of the airport has been optimized based on the flow and thermal results from air conditioning analysis using Autodesk CFD. With the change in the diffuser angles and CFMs as well as using mixed convection, hot spots with temperatures greater than 31^0 C have been reduced to the design temperature limit of 24^0 C.

The following results from Autodesk CFD Simulation provided insights into design adequacy:

- Air flow velocity values and pattern – a measure of air quality and cooling
- Temperature values and distribution – a measure of thermal comfort

Thus, air-conditioning analysis using Autodesk CFD led to the conclusion that optimized HVAC design is adequate for thermal comfort in passenger terminal building of airport.