

BES500169

Using Autodesk CFD for Human Comfort Ventilation Study for a Large Auditorium Hall

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

Learning Objectives

- Identify specific workflows in Autodesk CFD to simulate air flow and heat transfer for human comfort evaluation
- Implement workflows related to geometry, boundary conditions and results visualization
- Assess whether thermal comfort requirements are met
- Validate the design for adequacy of thermal comfort

Description

Design of large building spaces for human comfort is a challenge due to complex building geometry. Simple heat load based air-conditioning design may not suffice to ensure acceptable human comfort. Design based on such an approach would provide macro-level information without local variation and may lead to over design in general or design failure at specific locations. CFD simulation using Autodesk CFD provides a convenient way of evaluating human comfort throughout the building space. In this class, you will identify specific workflows in Autodesk CFD to simulate air flow and heat transfer in a large auditorium building. You will implement workflows related to geometry preparation, applying boundary conditions and visualizing airflow, temperature and thermal comfort parameters. Based on CFD results, you will assess if thermal comfort requirements are met or not. You will be able to validate the design for adequacy including any design change required before actual construction. dd your class description.

Speaker

Dr. Munirajulu. M, Bachelor of Technology (Hons.) and Ph.D. from IIT, Kharagpur, India, has more than 25 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 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, thermal comfort, 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.

AUTODESK UNIVERSITY

Introduction

Autodesk CFD is capable of simulating air flow leveraging mechanical ventilation to remove heat loads 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. The purpose of mechanical ventilation is to create *air movement*, driven by fans and provide *air changes* within the occupied space, replacing stale air with fresh air as well as removal of heat. This results in human comfort and good indoor air quality.

In Autodesk CFD, applications of mechanical ventilation include: *Thermal comfort of occupants within a building, data center cooling analysis, and smoke extraction analysis* for life safety.

Building

Building considered for CFD simulation is a large auditorium hall which is part of a large convention center called “India International Convention Center”, located in Delhi. Auditorium hall is multi use events area for plenary convention sessions, corporate banquets, corporate exhibitions, training, conferences etc. It has a seating capacity of 6000+ people in one volume over several levels

Auditorium Hall

Convention Center comprises of Main Auditorium hall, convention rooms and grand ball room with a total surface of 12830 sq.m. Main auditorium hall has a net surface of 5047 sq.m. Out of 5047 sq.m of surface area, 3048 sq.m are occupied by the seating area to accommodate 6000+ attendees. Due to this large occupancy, large amount of heat is generated in the auditorium hall which, if not removed will lead to occupant discomfort.



Convention Center

So, the goal of air-conditioning system design must provide occupant thermal comfort during events held in the auditorium hall.

Autodesk CFD Workflows for Thermal Comfort

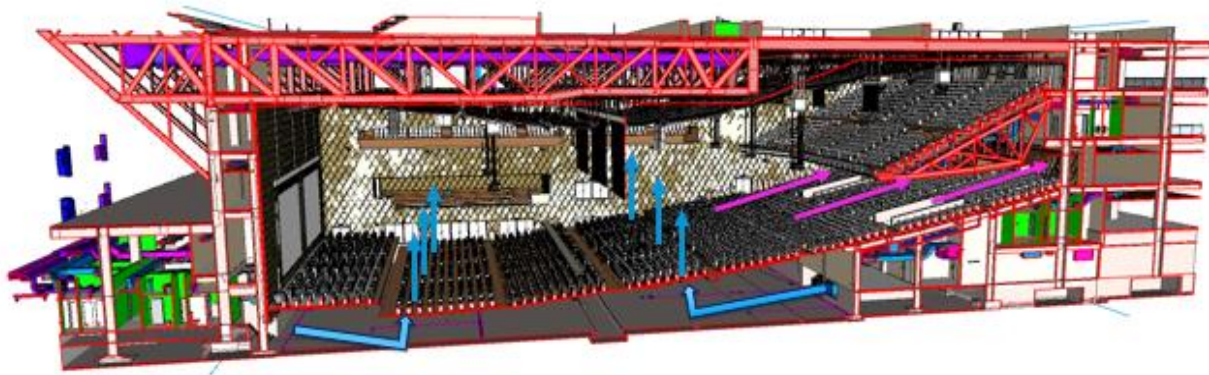
Ensuring thermal comfort to occupants is critical for hosting events that will attract contributed income to keep the venue full and active with financial success year after year.

Auditorium Hall Geometry

Main auditorium hall is a large building. It is 74m in length, 23 m in height at the stage and 14 m height at the balcony level. Auditorium has a width of 14 m to accommodate seating rows both in the main hall as well as balcony. Air-conditioning system is a vital part for use of auditorium. Air-conditioning system in this case uses displacement ventilation. Cool air is introduced at low level and as such ventilates the occupied zone of the volume. Air is supplied at low level via outlets incorporated into the seating with a low temperature differential. Air is supplied at low velocity and rises as it is warmed by heat gains within the space. Extract is from high level to ensure the ventilation strategy complements thermal buoyancy rather than fighting against it.

Displacement Ventilation

Displacement ventilation gives a better air quality as indoor air is pushed out of the occupied zone. The displacement ventilation system returns stale air at high level to the air-handling units where it is passed through heat recovery. Return air is taken from a higher stratified level outside the occupied zone. The heat recovery system within the air-handling units operates by extracting the “cold” from the return air (which although warmed by occupants and other heat gains is cooler than the outside fresh air would be) and pre-cooling the supply air stream.



Air-Conditioning System – Auditorium Hall

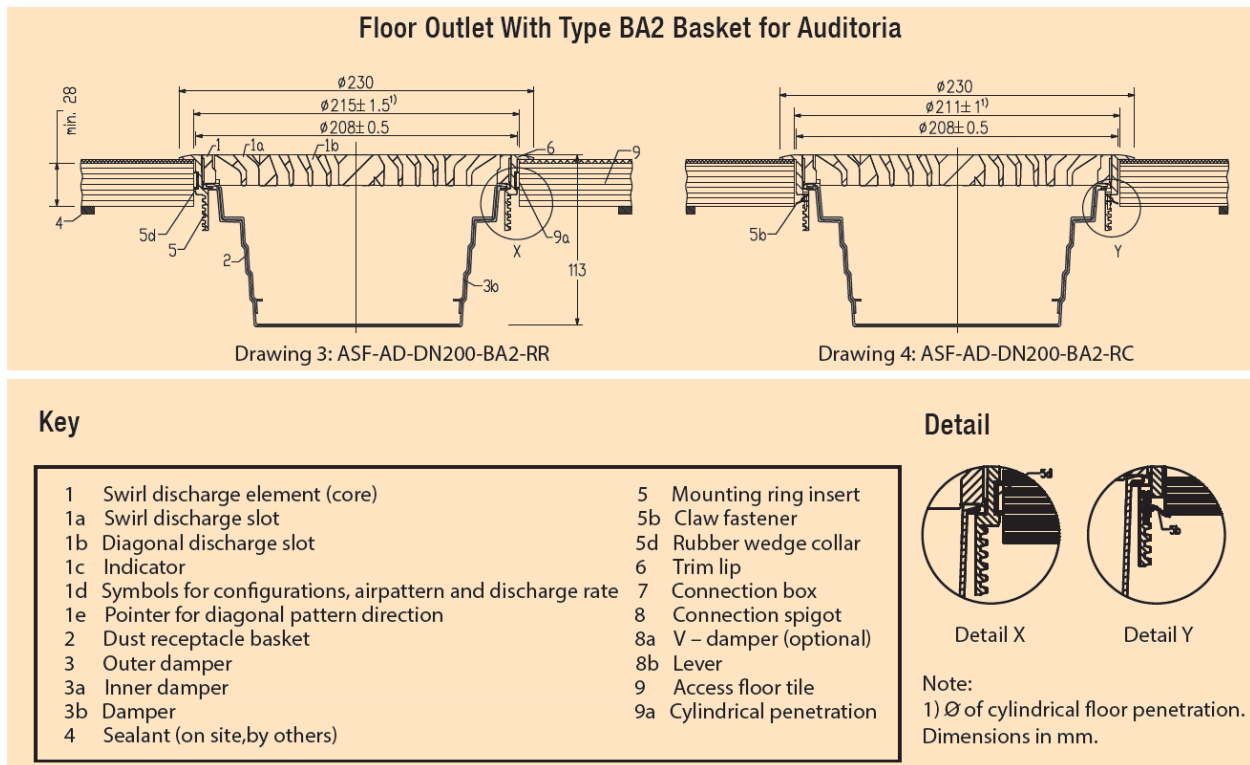
Component Characterisation

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.

Characterisation is a strategy to capture design intent with minimum amount of necessary complexity, to optimise simulation performance in terms of ability to mesh, reduce solver time and produce valid results.

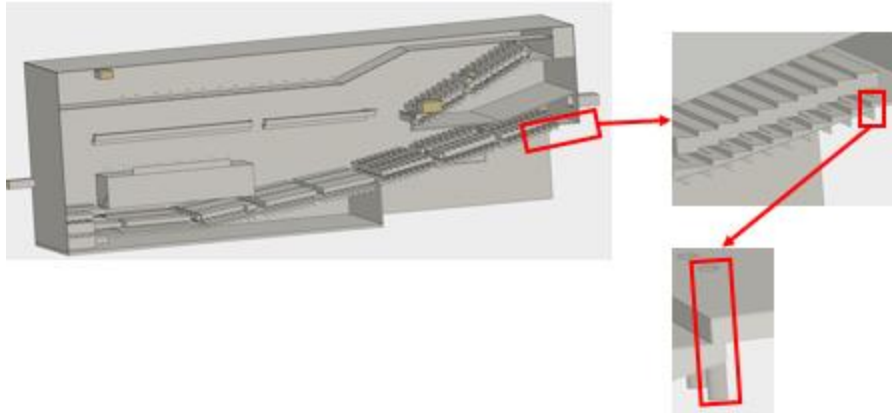
While a fully detailed building model is necessary for construction, most of the geometric complexity may not be required when leveraging simulation for obtaining reliable performance insight quickly and effectively, warmed by occupants and other heat gains is cooler than the outside fresh air would be) and pre-cooling the supply air stream.

From the CAD geometry of auditorium hall, the model suitable for CFD simulation is prepared. Input details for CFD analysis include building geometry from CAD data including seating, air-conditioning scheme details such as supply diffusers and outlets for return air



Air-Conditioning System – Supply Diffuser Physical Model

Actual supply diffuser units will have a number of details such as swirl discharge element, swirl discharge slots, diagonal discharge slots etc. It would be practically cumbersome to model individual components with all the details because it would increase mesh count and compute time. However, for system level performance, which is our goal in this project case study, actual diffuser geometry is represented in the Autodesk CFD model as a simple geometrical feature.

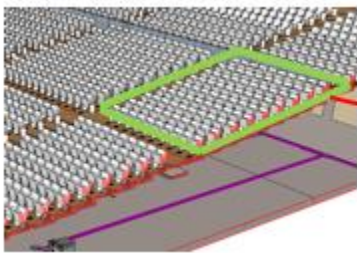


Air-Conditioning System – CFD model characterization

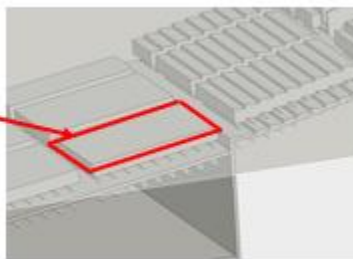
The diffuser is modelled as cylindrical body with circular opening representing the shape of the diffuser. The outlet face is provided with resistance value of 50 %, meaning 0.5 or 50% (free area ratio) to depict the diffuser opening.

The occupant seating and occupants are modelled as block of air volume, due to the fact that supply diffusers are integrated to the seating.

It is important to assign heat gains to the occupant volume in order to evaluate air temperature. Heat gains include people load, equipment and lighting loads. Seated height is considered as 1.1 m from the floor.



CAD model



CFD model characterization

Set up Tasks

The set-up tasks of simulation for materials, boundary conditions, meshing depend on and applied to the input geometry. Each volume or surface created in CAD is selectable in Simulation CFD. Selectable items are used to define inputs and extract results.

CAD model for CFD is created by modifying existing CAD data or create a new geometry for simulation by referencing the existing geometry and it is a great way to develop a clean simulation model that captures design intent. This strategy is often less time consuming than modifying and troubleshooting existing data. Modified CAD data is loaded into Autodesk CFD.

AUTODESK UNIVERSITY

Materials

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.

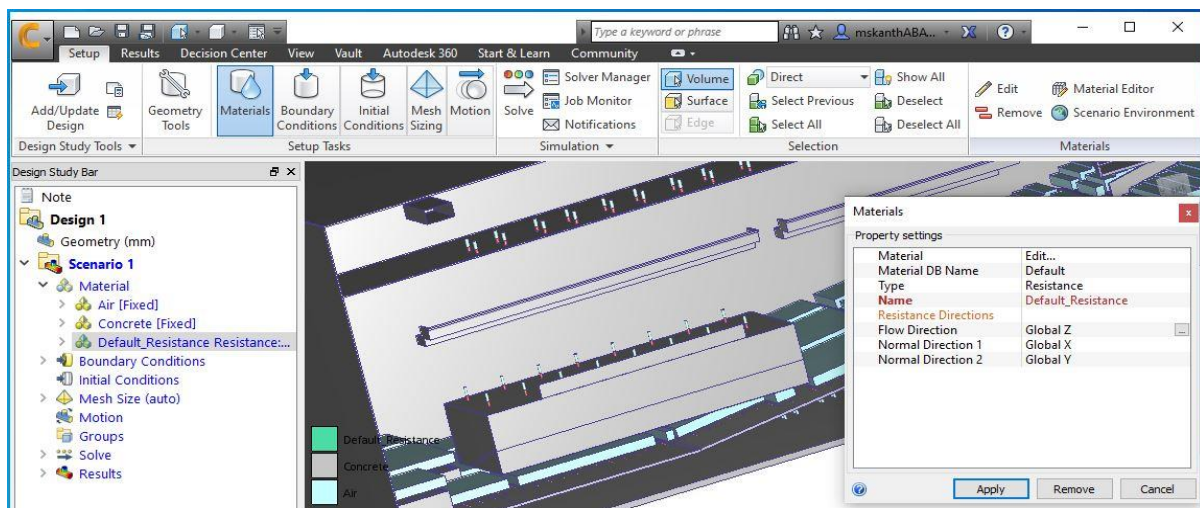
For AEC applications, “air” is by far the most common fluid material. For Auditorium CFD simulation, fluid domain is assigned “air” as material with fixed properties (i.e., density does not vary with temperature)

Other materials used include solids for building enclosure and these solids are suppressed from the simulation.

The default resistance is provided to give resistance to the flow with a free area ratio of 0.5 in the supply inlets/diffusers.

The structural elements in the building are assigned as “solid” with concrete material

Humans are assigned as air volume with heat generation assigned in the boundary condition workflow.



Materials assignment

Boundary Conditions

We specify boundary conditions after assigning materials in CFD model. Boundary Conditions (BCs in short) are necessary inputs to produce a valid flow and thermal solution. Defining these quantities is critical to a converged solution in Simulation CFD. The simulation environment interacts with the simulation model based on BCs.

Pressure BC is applied on the surface of the air volume extension on the outlet.

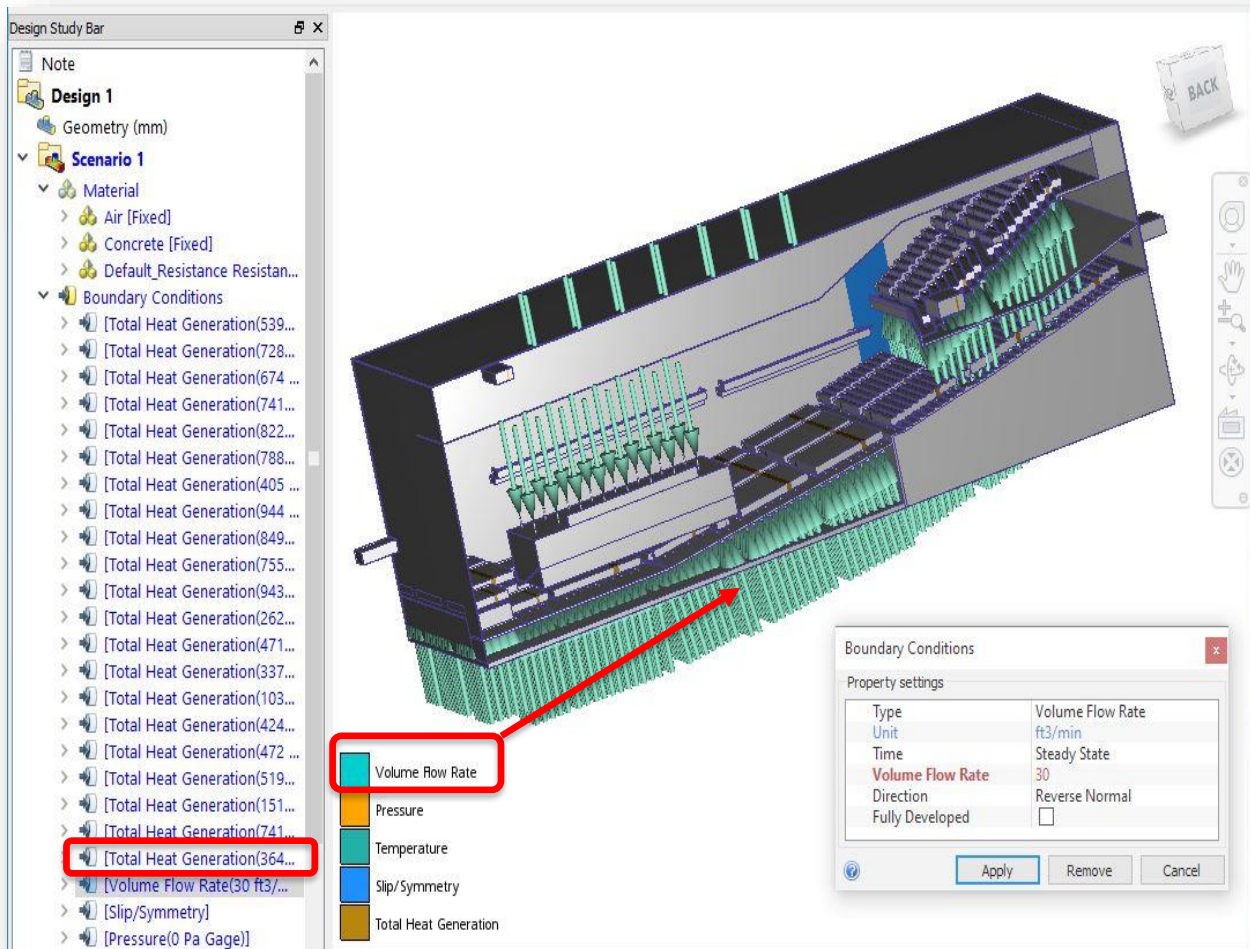
Inlets are extended by at least a hydraulic diameter to avoid flow recirculation.

Air is mechanically moved out of the building space, so inlets are assigned volume flow rate on each supply/inlet diffuser.

AUTODESK UNIVERSITY

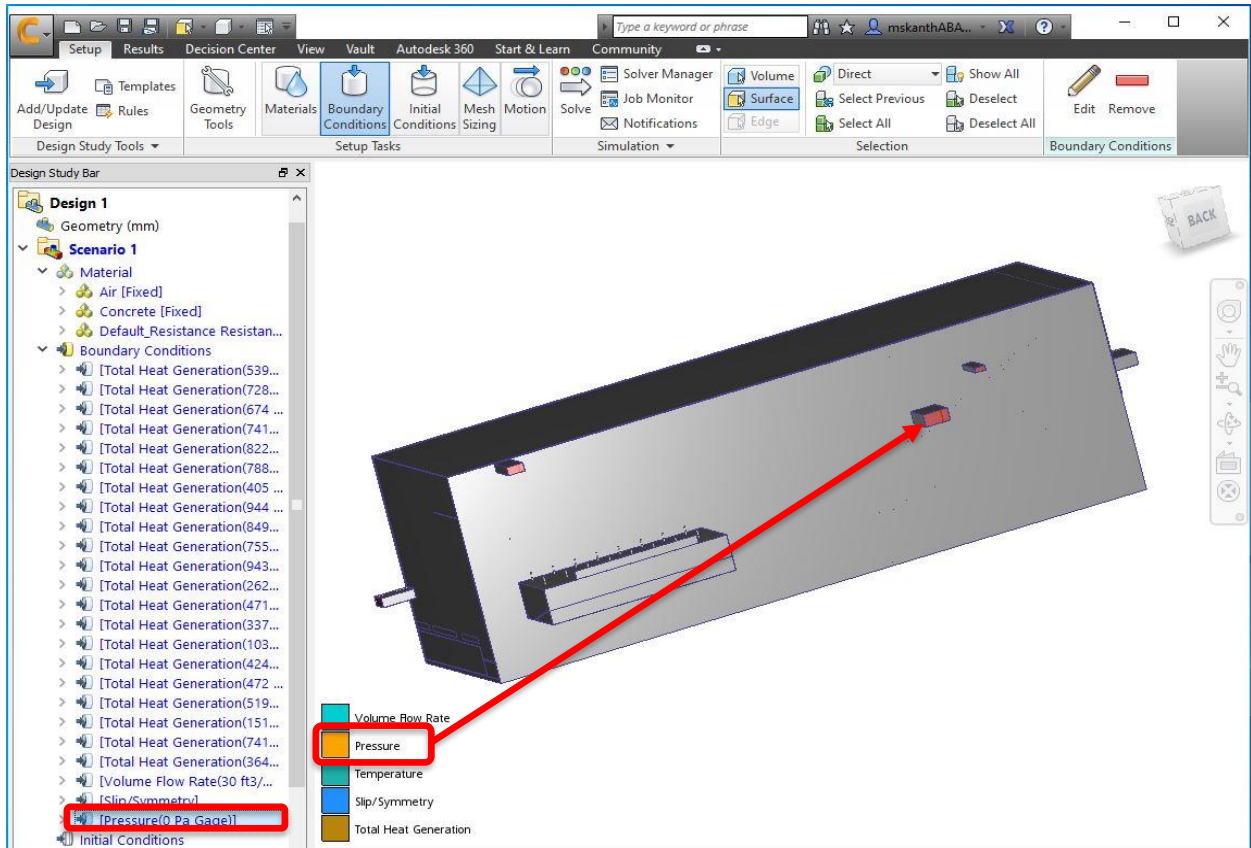
Arrow direction indicates the flow in the right direction from supply/inlet diffusers i.e., direction is into the auditorium space.

For Auditorium hall, thermal simulation is driven by thermal boundary conditions and deals with heat generation components such as people, equipment and lighting and so that is taken care by defining total heat generation boundary condition for occupant volume.



Inlet Boundary Condition – 30 CFM through each supply diffuser

AUTODESK UNIVERSITY



Outlet Boundary Condition – Zero-gauge pressure

The screenshot displays the Autodesk CFD interface. On the left, the 'Design Study Bar' shows a tree view for 'Design 1' with 'Scenario 1' expanded to 'Boundary Conditions'. A red circle highlights a list of 20 'Total Heat Generation' conditions with various values. A red arrow points from this list to a 'Boundary Conditions' dialog box on the right. The dialog box shows the following property settings:

Property settings	
Type	Total Heat Generation
Unit	W
Time	Steady State
Temperature Dependent	Disabled
Total Heat Generation	8493

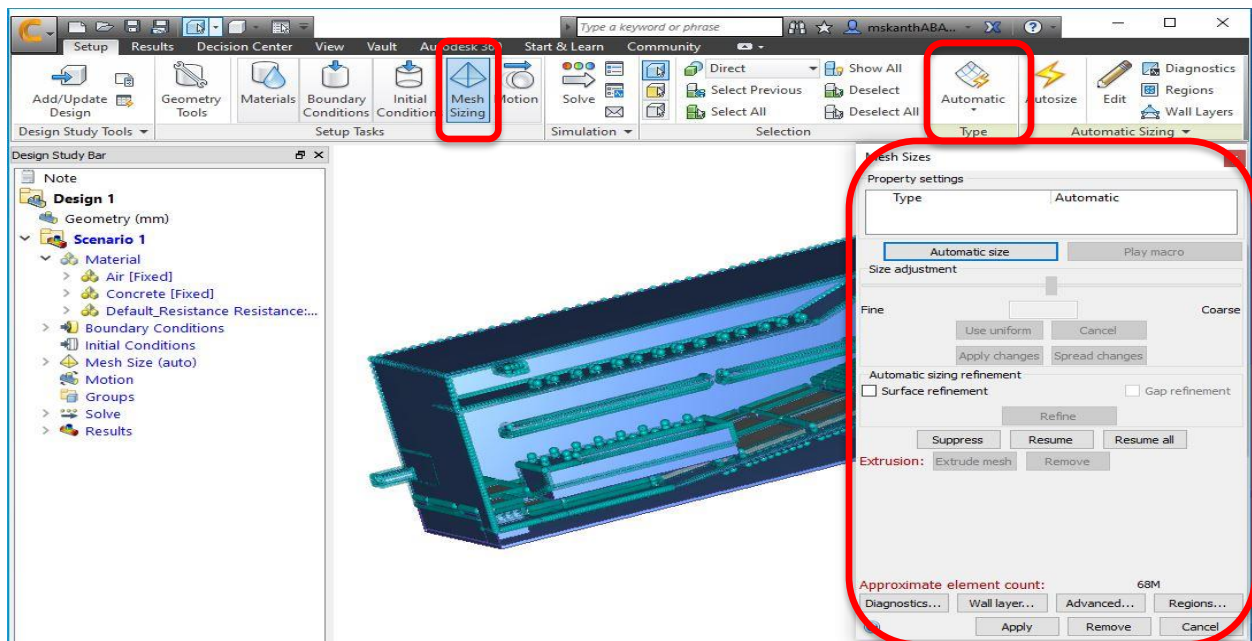
At the bottom of the dialog box are 'Apply', 'Remove', and 'Cancel' buttons. The status bar at the bottom of the software window indicates 'On Plane 2- Location(X=153386,Y=22589.7,Z=-5651.93)- Value 20.5024 Celsius' and '5 Volume(s) selected'.

Heat Load Boundary Condition

Meshing

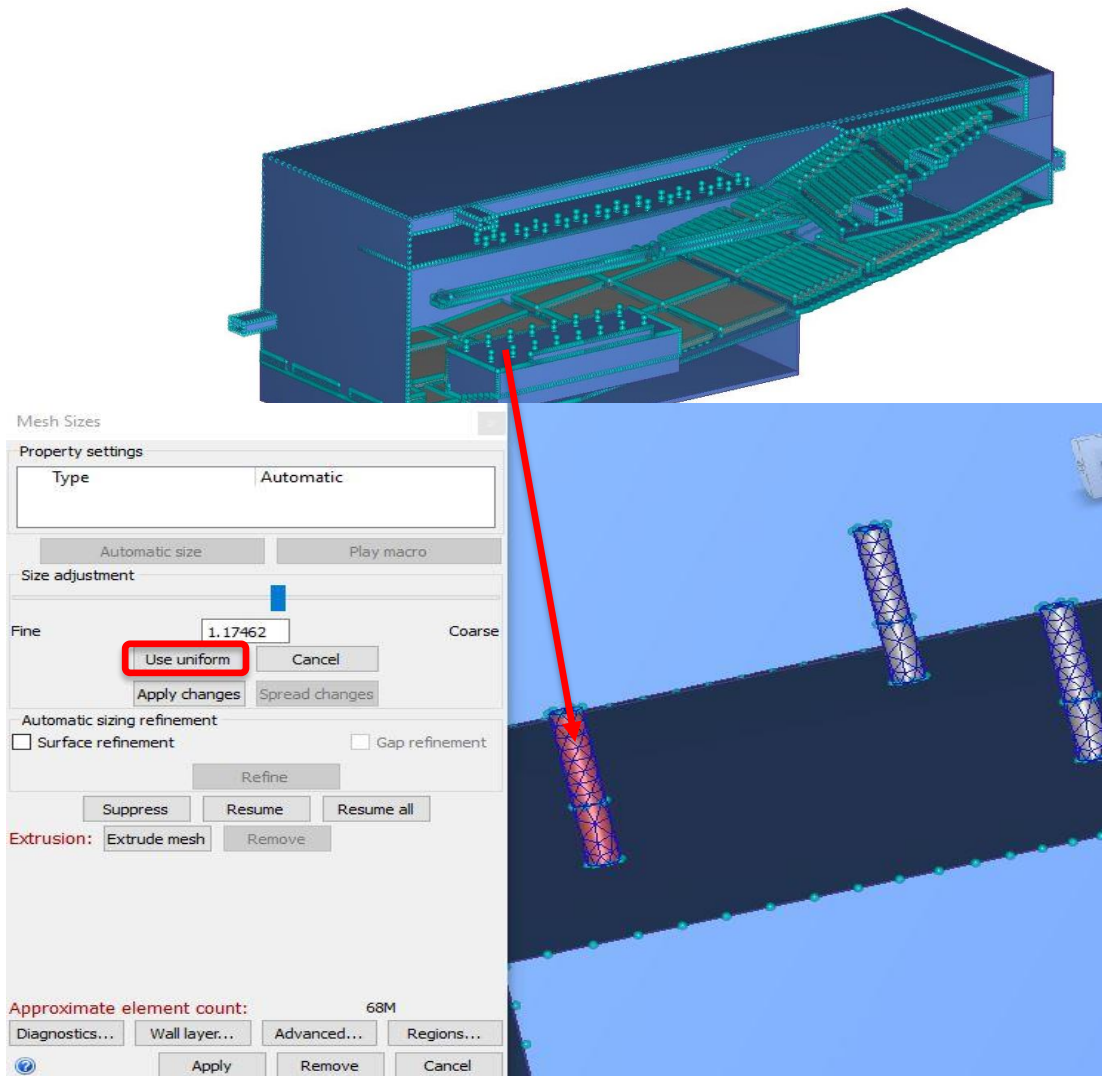
Automatic mesh sizing is used to define the mesh distribution in the CFD model domain. CFD simulation uses FEM to calculate the fluid flow and thermal results. The FE mesh is the backbone of CFD simulation/calculation and has a direct impact on solution accuracy. In meshing task, the model geometry is 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 for mesh independent solution. Here for our auditorium simulation, we have gone up to 16 million elements. For better accuracy, first we do automatic mesh sizing for the entire domain and then we do refinement on volumes as necessary.



Automatic Meshing

Local mesh refinement with uniform mesh is used for mesh on supply diffusers and occupant volumes to capture the flow through them.



Simulation and Results Visualisation

The solve dialog in the simulation task is the command center for the simulation. It controls the physics of simulation as well as what data is to be output.

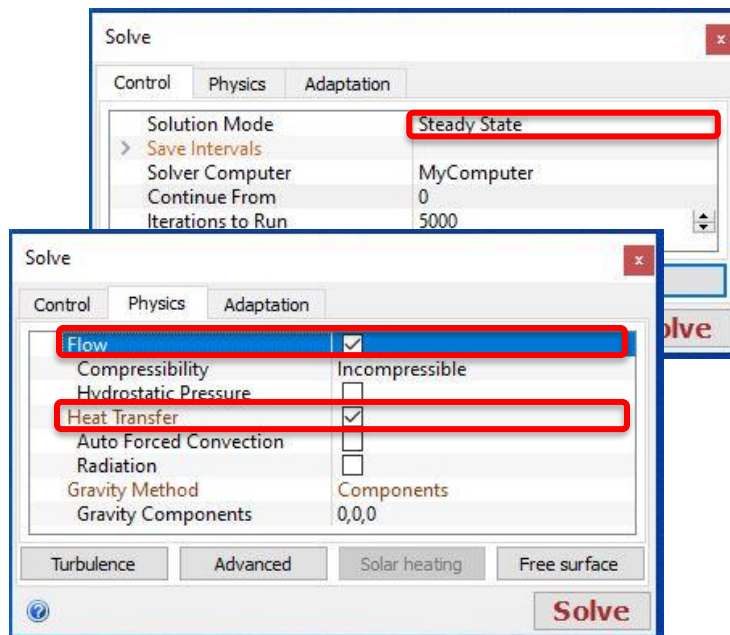
Results dialog in the simulation CFD controls various methods for visualising results i.e., results using Global, Plane, Traces, ISO Surfaces and ISO Volumes option

Solver settings are based not only on the physics of the application but also user preferences such as how much 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 to knowing which settings are required is to determine the physics of the problem being solved.

Solver Settings

Flow is solved with steady state, *incompressible* flow option. *Heat transfer* needs to be considered in this analysis, so heat transfer is enabled in the solution settings as shown in the image here. For thermal comfort study, we need to calculate temperature i.e., we have to solve flow as well as heat transfer. Buoyancy effects due to gravity need not be considered. So, heat transfer is by forced convection only.



Solver settings

CFD Results

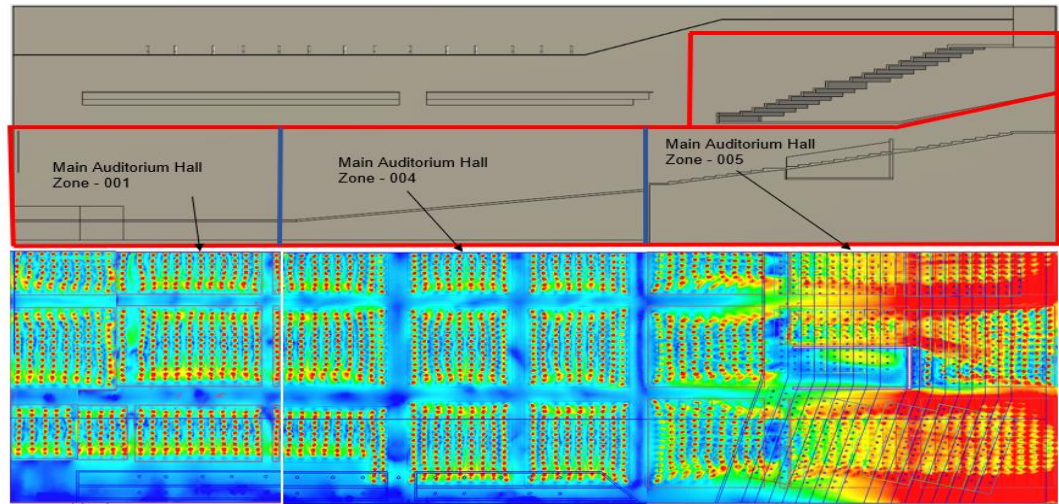
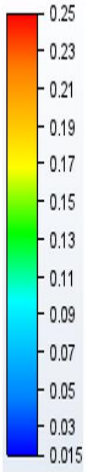
After simulation is set up, run and converged, we can look at results using “Results Visualization” tools. Autodesk CFD simulation helps the designer to validate or optimize the design by providing visualization of performance characteristics such as temperature, velocity that are difficult to capture in the real world at all locations.

Anemometers, manometers, thermocouples and infra-red cameras can be used to capture performance characteristics at specific location and time. So physical testing would give only limited depiction of performance characteristics. Simulation CFD extends limitations of physical testing by providing needed values everywhere in the building so CFD analysis results are useful to evaluate performance, identify opportunities to improve and effect of design modifications on performance.

Flow and thermal performance of auditorium can be evaluated by looking at results using global option.

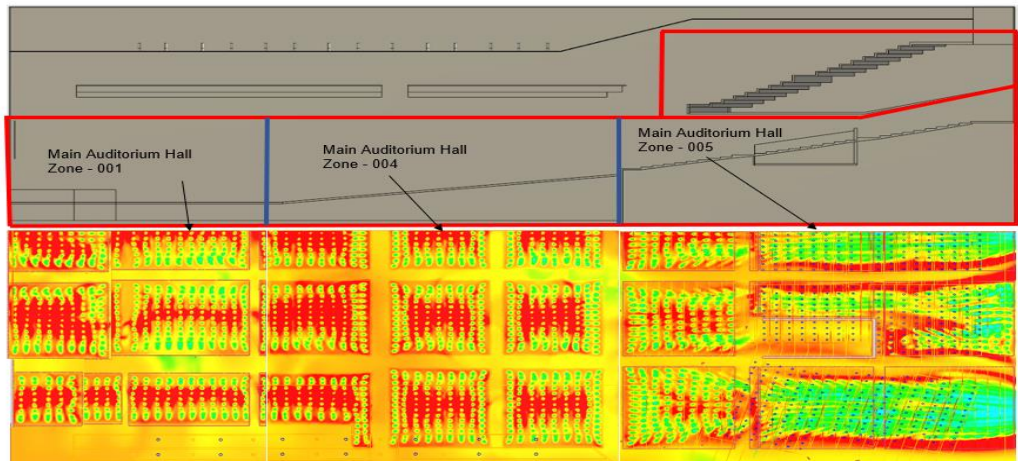
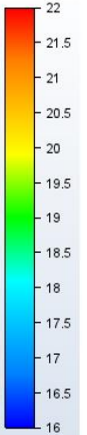
Global results for temp provide visualization of thermal gradients on the model surfaces, showing the minimum and maximum temperature in the analysis

Velocity m/s



Velocity at 1.7 m level

Temperature °C

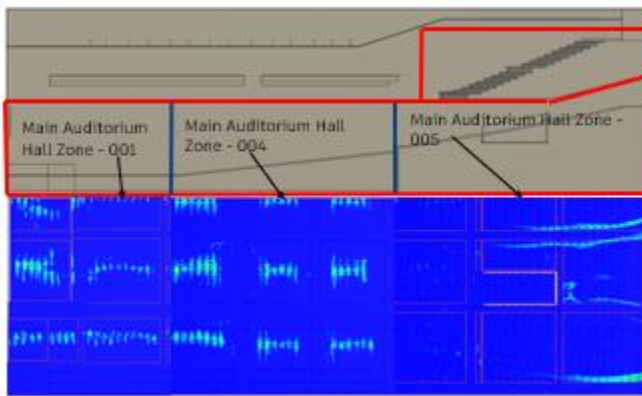
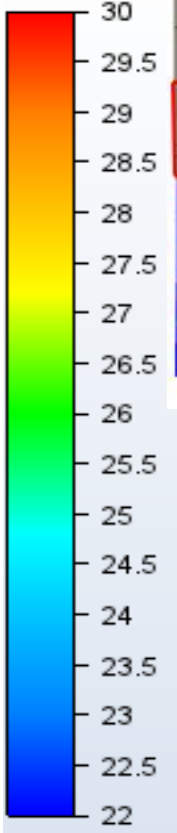


Temperature at 1.7 m level

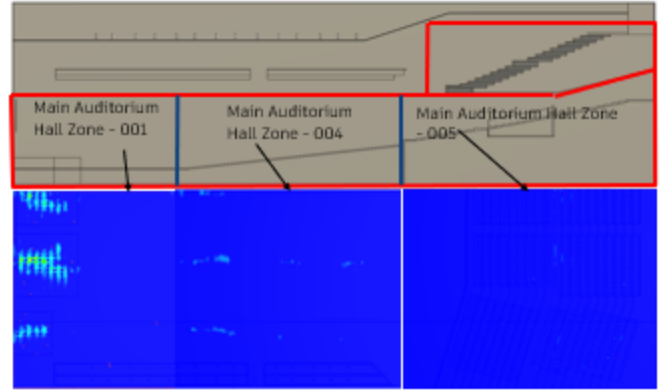
Air flow velocity and temperature values are the key results from CFD analysis to evaluate thermal comfort. Results Plane option in this image provides temperature as well as velocity distribution in the occupied space.

AUTODESK UNIVERSITY

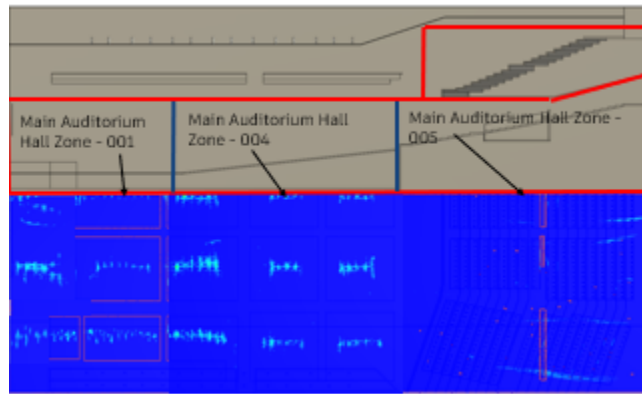
Temperature °C



Temperature contour at 0.7 m from the floor

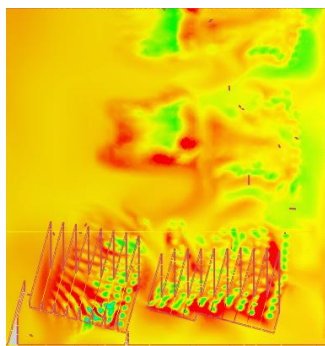
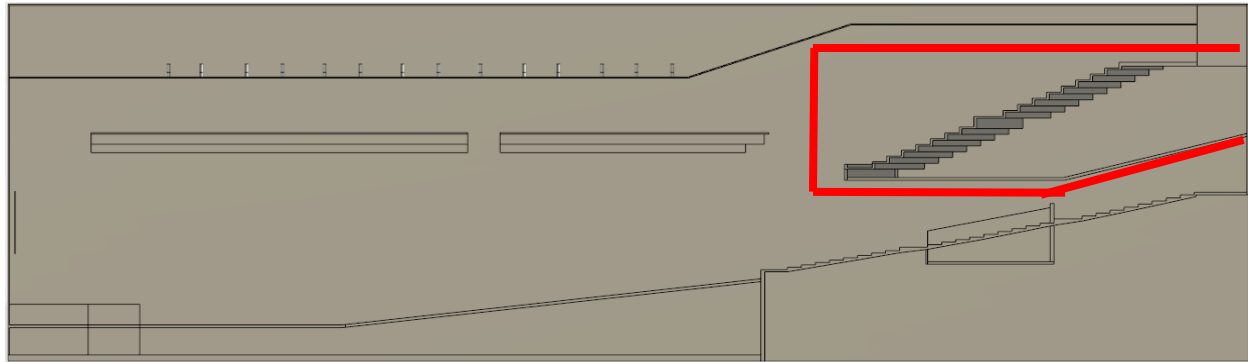
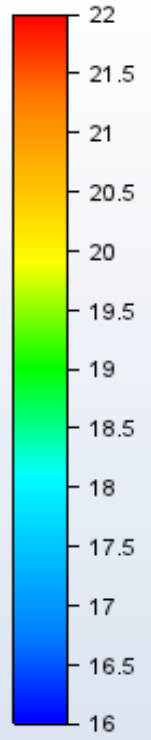


Temperature contour at 1.7 m from the floor

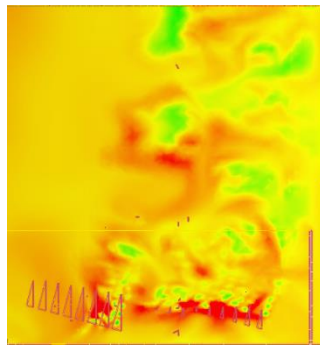


Temperature contour at 1.1 m from the floor

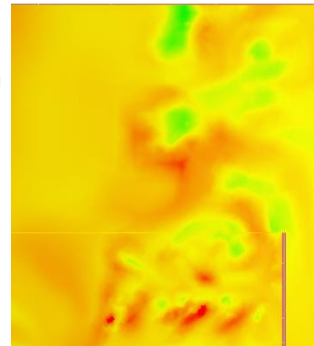
Temperature values plotted in the range of 21 to 30° C



Temperature contour at 0.7 m from the floor

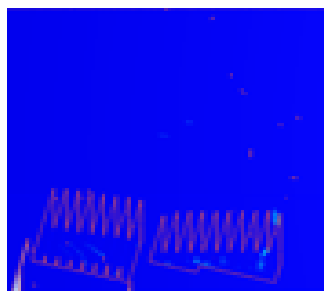
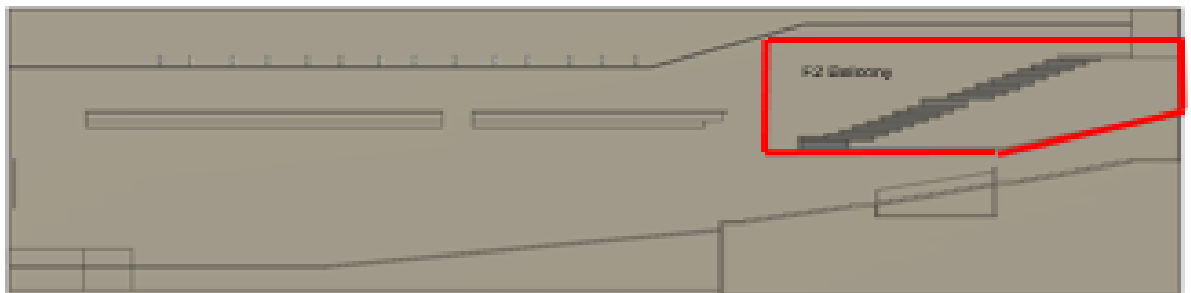
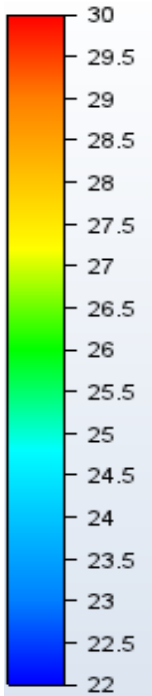


Temperature contour at 1.1 m from the floor

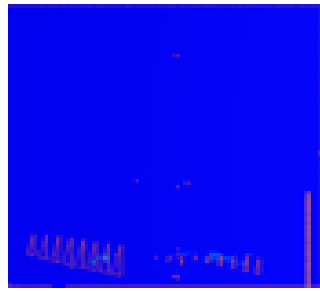


Temperature contour at 1.7 m from the floor

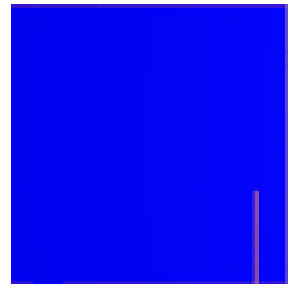
F2 Balcony Temperature values plotted in the range of 16 to 22^o C



Temperature contour at 0.7 m from the floor



Temperature contour at 1.1 m from the floor



Temperature contour at 1.7 m from the floor

F2 Balcony Temperature values plotted in the range of 22 to 30^o C

Review Results and Evaluate Design

Based on CFD results of temperature distribution and values at occupied level, we can evaluate if air-conditioning system design is adequate or not for thermal comfort.

We have seen that from relevant CFD results, temperature profiles are within acceptable limits for occupant thermal comfort, thereby indicating confidence in the adequacy of air-conditioning system design.

The velocity contour over the Main Auditorium Hall shows a uniform distribution of airflow. The Temperature distribution in the Main Auditorium shows most of the occupied area is within the recommended value of 22° C.

The Temperature distribution in the F2 Balcony shows that most of occupied area has temperature in the range of 19° C – 21.5° C which is acceptable. Hence proposed air-conditioning system can be considered to be adequate.

To wrap up, we have covered in this session the following and hope you have benefited from these key learnings which are:

- Identifying specific workflows in Autodesk CFD
- Implementing these workflows
- Assessing CFD results
- Validating the design for thermal comfort

Thank you and happy learning at AU2021.....