

# Autodesk CFD for Thermal Comfort in Auditorium Hall

**Dr. Munirajulu. M**

HEAD - CFD | @m\_munirajulu

# About the Speaker

Dr. Munirajulu. M, Larsen & Toubro Limited

**Passionate about performance-based design and use of simulation tools in design of buildings.**

- 25+ years industry experience in CFD analysis
- Using Autodesk CFD Simulation for MEP design
- **Focus areas:** Data center cooling, Basement car park ventilation, DG room ventilation, Smoke simulation in buildings, Air-conditioning and thermal comfort analysis for large and complex buildings
- Speaker at AU 2017, 2018, 2019, 2020 US & AU2019 India



# Role of Autodesk CFD Simulation

## Performance based design

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
- Lack of local variation
- Over design /design failure

Alternatively, CFD simulation using Autodesk CFD provides a convenient way of evaluating human comfort throughout the building space.

- 3-D analysis / virtual design
- What-if scenarios
- Design effectiveness

# Key Objectives

# Key Learning Objectives

- Identify specific workflows in Autodesk CFD
- Implement workflows
- Assess CFD results
- Validate design for thermal comfort

# Building – Auditorium Main Hall

Largest Hall in Asia

Auditorium hall is the principal component of events area in the convention center.

- Main component of events area
- 6000+ seater capacity
- Venue for plenary sessions, state/catered dinners, training, conferences, trade/corporate/cultural presentations

Auditorium has flexible design such that it can seat 6000+ people in one volume over several levels or be divisible into two halls of 4000 and 2000 capacity each by means of a vertical retractable/foldable acoustic partition.

*Incredible India*

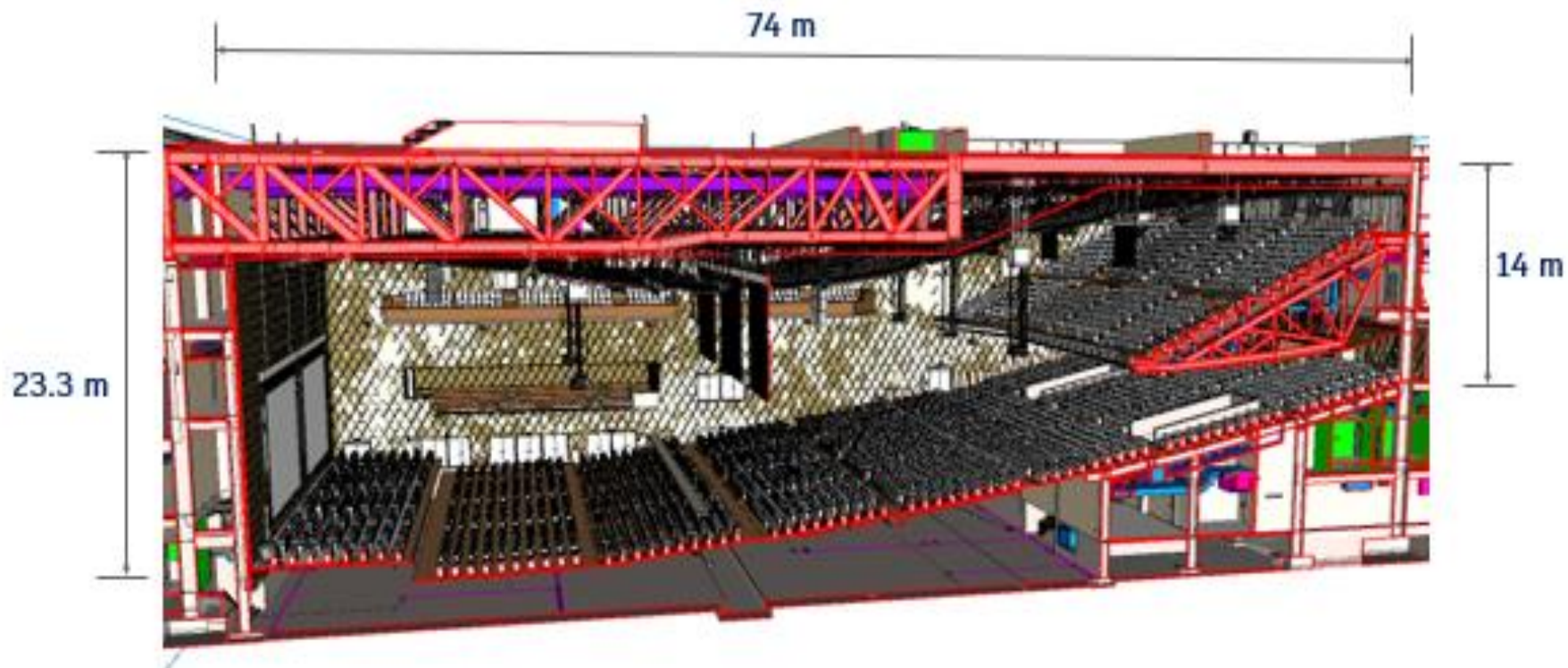


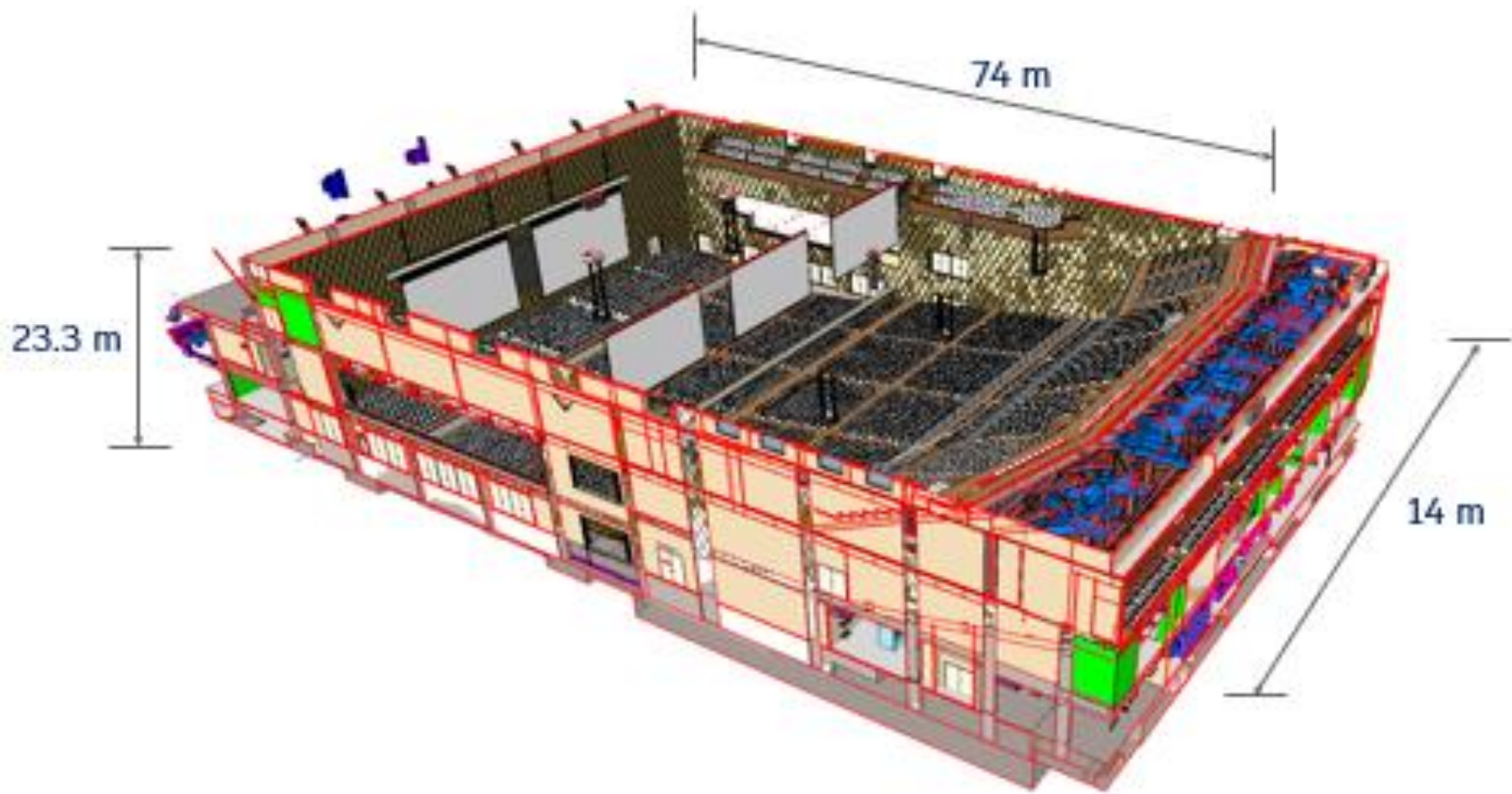




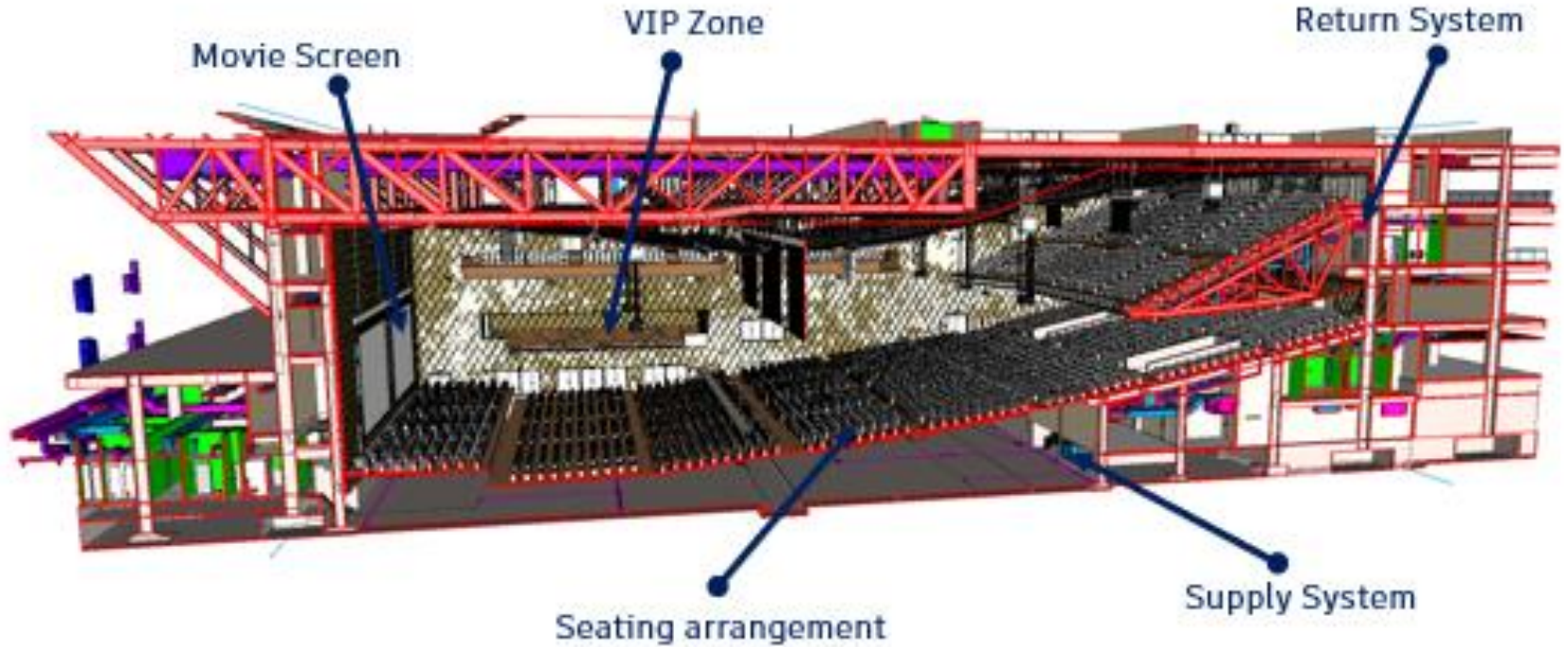
# Thermal Comfort

# Auditorium Main Hall Geometry

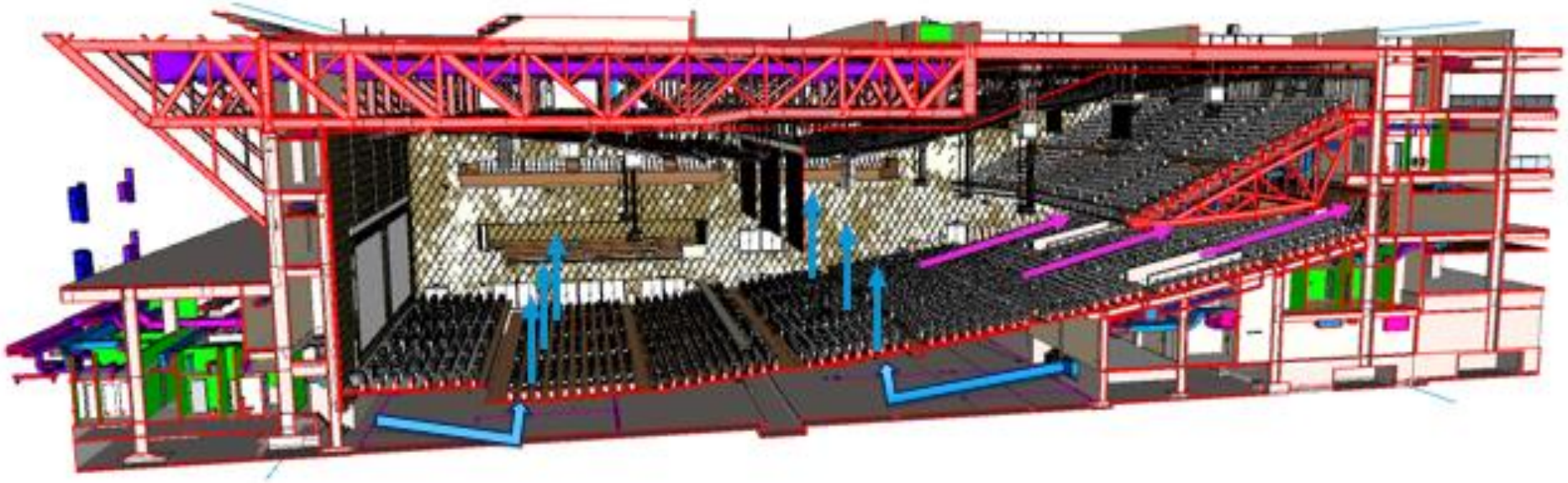




# Air-Conditioning System



# Air-Conditioning System



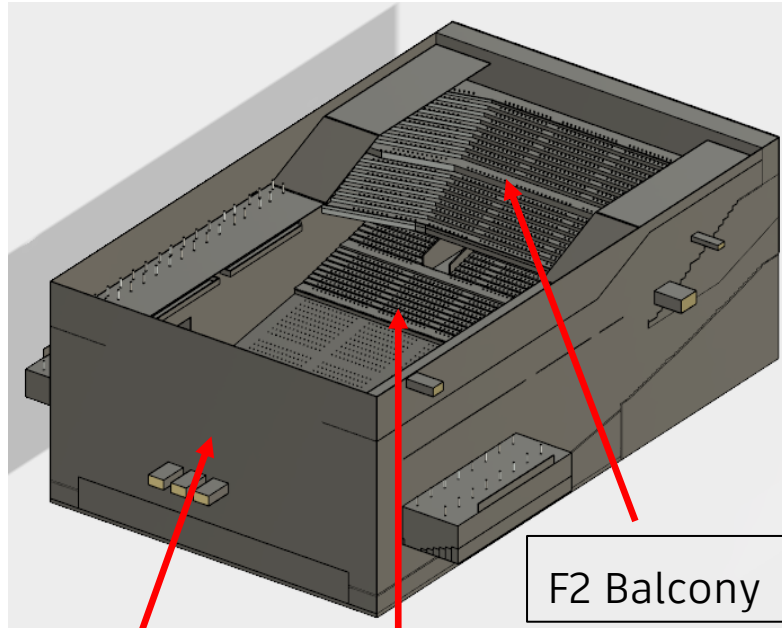
Conditioned air supplied through  
diffusers below the seat

Return air is drawn through various vents  
provided in the sides and back of the  
auditorium hall



# **Component Characterization**

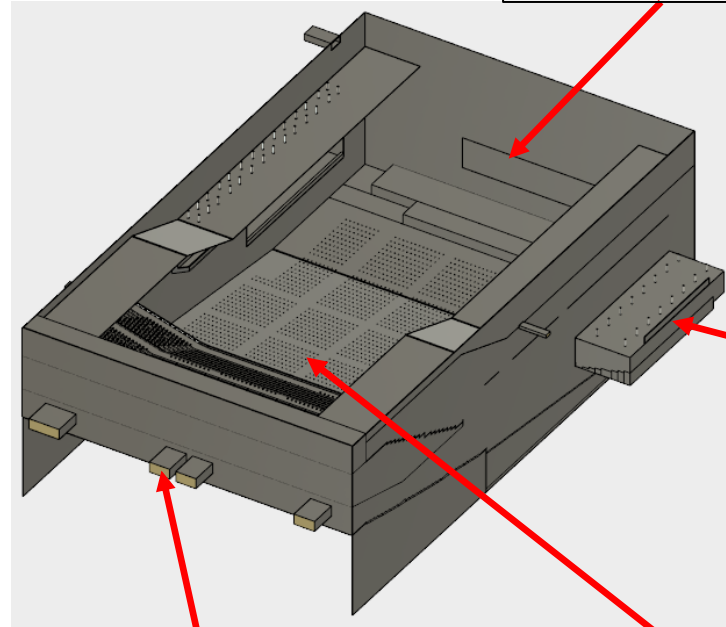
# Auditorium Hall CFD Model



F2 Balcony

Main Auditorium Hall

Auditorium Screen



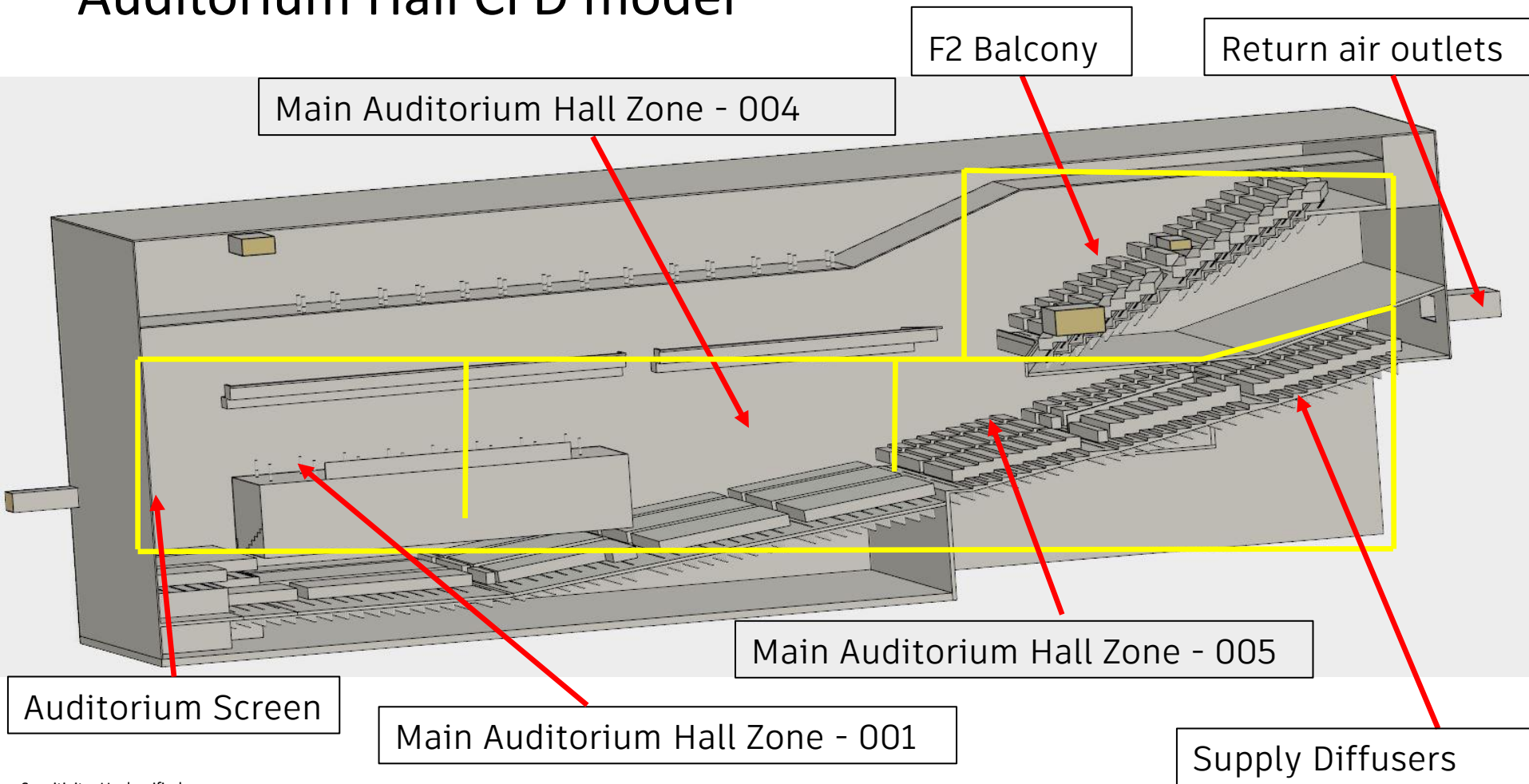
Auditorium Screen

VIP Box

Exhaust Outlets

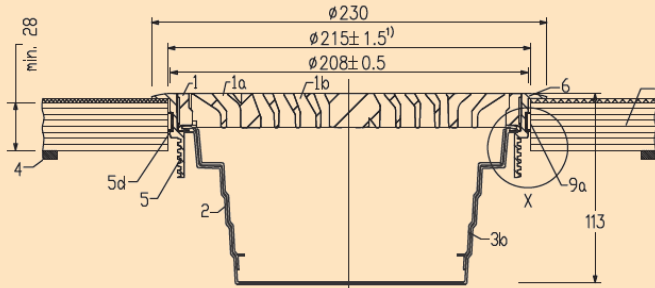
Supply Diffusers

# Auditorium Hall CFD model

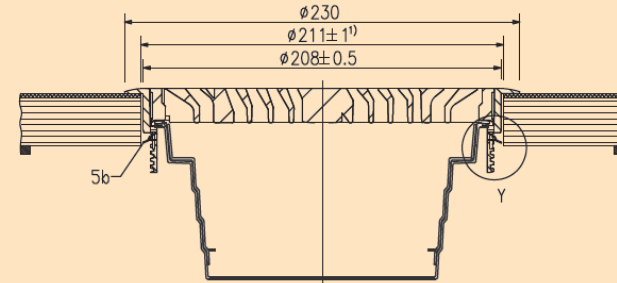


# Auditorium Hall Supply Diffuser Specifications

## Floor Outlet With Type BA2 Basket for Auditoria



Drawing 3: ASF-AD-DN200-BA2-RR

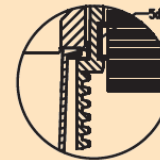


Drawing 4: ASF-AD-DN200-BA2-RC

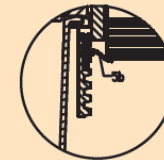
### Key

- |  |                            |
|--|----------------------------|
| 1 Swirl discharge element (core)                             | 5 Mounting ring insert     |
| 1a Swirl discharge slot                                      | 5b Claw fastener           |
| 1b Diagonal discharge slot                                   | 5d Rubber wedge collar     |
| 1c Indicator   | 6 Trim lip                 |
| 1d Symbols for configurations, airpattern and discharge rate | 7 Connection box           |
| 1e Pointer for diagonal pattern direction                    | 8 Connection spigot        |
| 2 Dust receptacle basket                                     | 8a V – damper (optional)   |
| 3 Outer damper   | 8b Lever                   |
| 3a Inner damper  | 9 Access floor tile        |
| 3b Damper  | 9a Cylindrical penetration |
| 4 Sealant (on site, by others)                               |                            |

### Detail



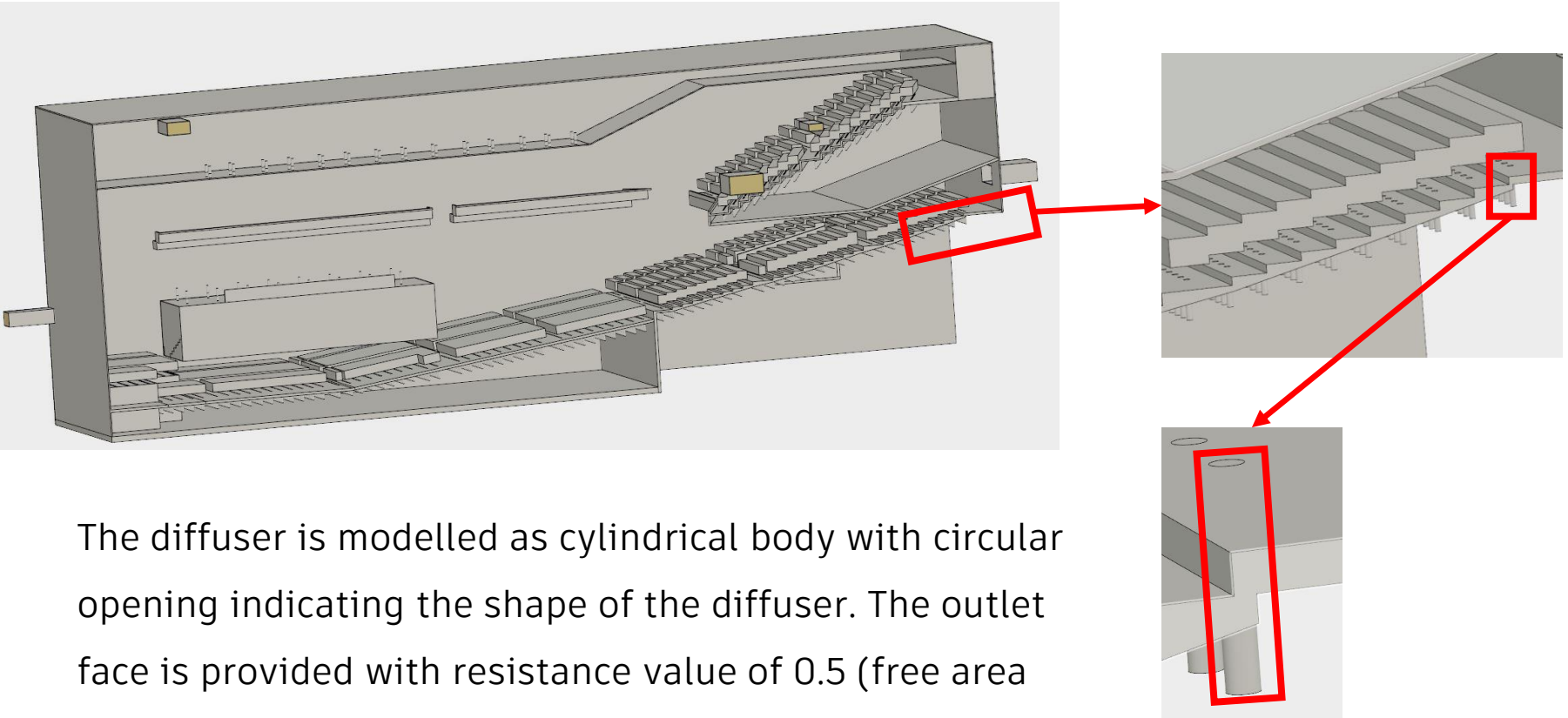
Detail X



Detail Y

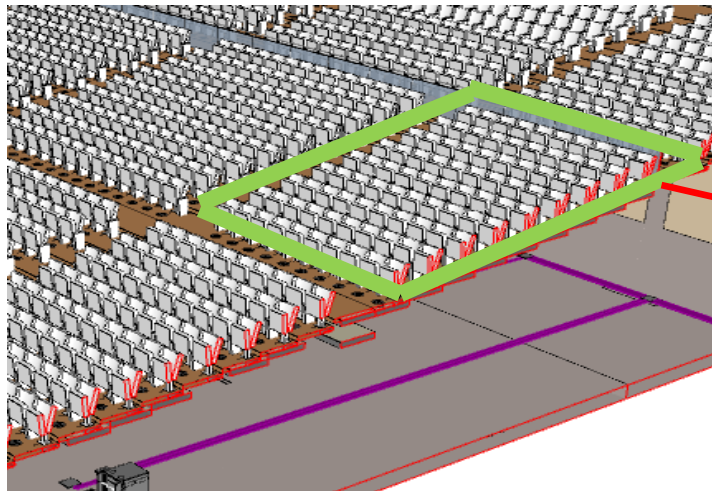
Note:  
1)  $\varnothing$  of cylindrical floor penetration.  
Dimensions in mm.

# Auditorium Hall Diffuser Characterization

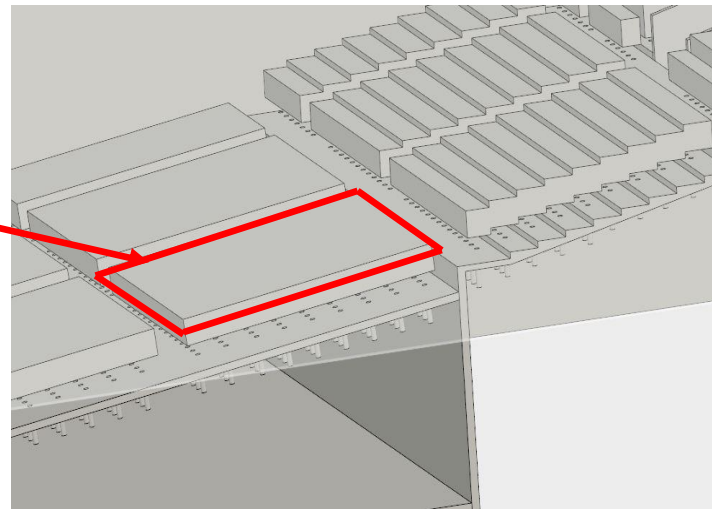


The diffuser is modelled as cylindrical body with circular opening indicating the shape of the diffuser. The outlet face is provided with resistance value of 0.5 (free area ratio) to depict the diffuser opening.

# Auditorium Hall Human Occupant Characterization



CAD model



CFD model

The occupant chairs are modelled as block of air volume representing occupants as shown by the CFD model.

# Auditorium Hall Design Details

S.No	Floor Region	People Capacity
1.	Main Auditorium Hall	4264
2.	F2 Balcony	1148
3.	VIP Box	112

1.	Lighting load	0.7 W/ft <sup>2</sup>
2.	Equipment load	1.0 W/ft <sup>2</sup>
3.	Sensible heat load	67.4 W/person
4.	Seated height	1.1 m
5.	Humidity	50%
6.	Inlet air temperature	16°C



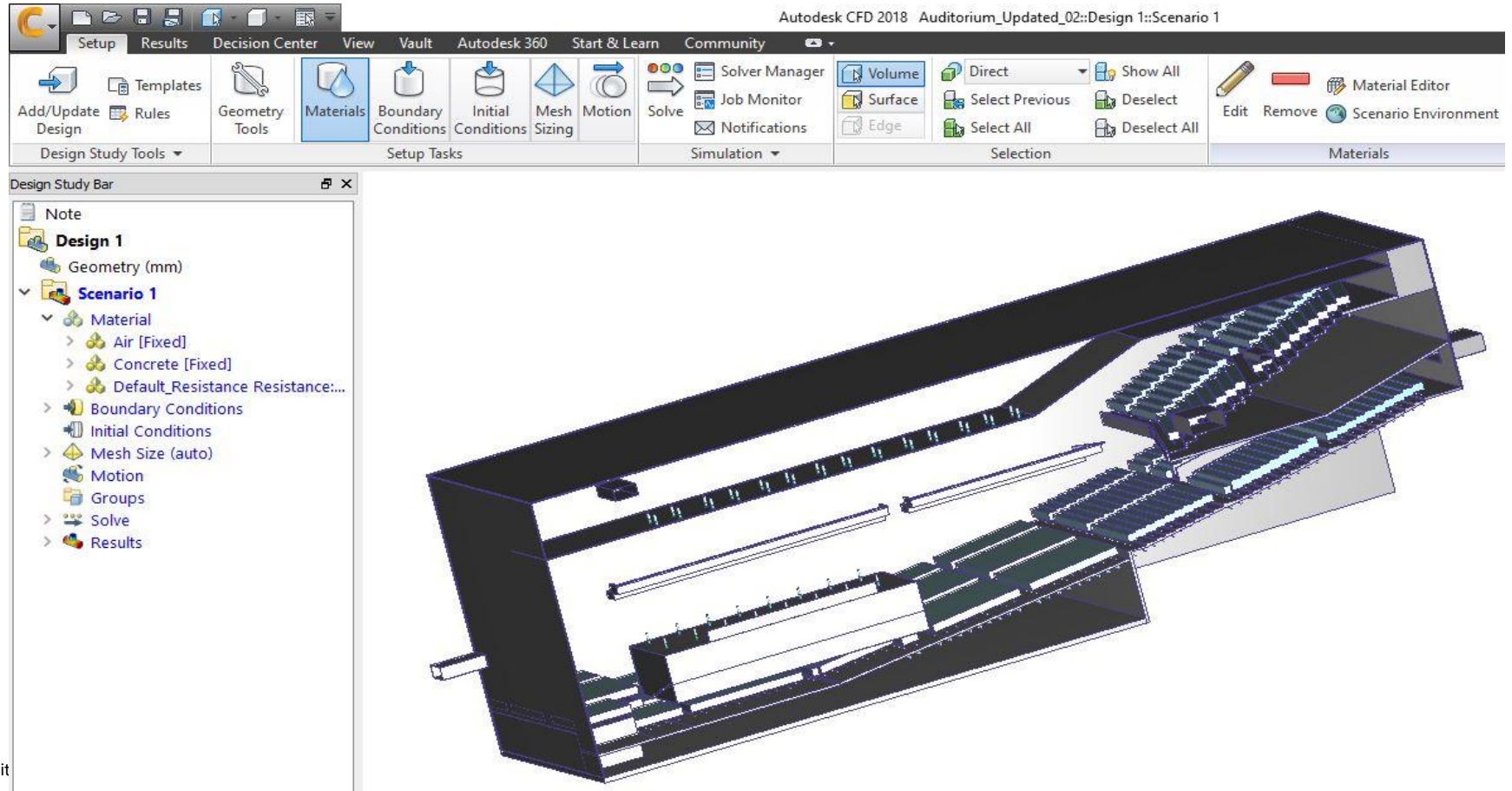
# **Modeling Strategies- Airflow and Heat Transfer**

# Autodesk CFD Set up Workflow

The screenshot displays the Autodesk CFD software interface. The top ribbon is divided into several tabs: Setup, Results, Decision Center, View, Vault, Autodesk 360, Start & Learn, and Community. The Setup tab is active, and a red box highlights the 'Setup Tasks' section, which includes icons for Geometry Tools, Materials, Boundary Conditions, Initial Conditions, Mesh Sizing, and Motion. Other sections of the ribbon include Solver Manager, Job Monitor, Notifications, Simulation, Selection, and Materials. The Design Study Bar on the left shows a tree view with 'Design 1' expanded to 'Scenario 1', which includes 'Material' (set to 'Unassigned'), 'Boundary Conditions', 'Initial Conditions', 'Mesh Size', 'Motion', 'Groups', and 'Solve' (with 'Flow: On' and 'Heat Transfer: Off' options). The main workspace shows a 3D coordinate system with a green line and a red line, with numerical values at various points: 555.58, 416.685, 277.79, 138.895, 515.906, 1,031.81, 1,547.72, 2,063.62, 1,168.16, 1,752.24, and 2,336.32.

- CAD model
- Material assignment
- Boundary Conditions
- Meshing

# Auditorium Hall CFD Model

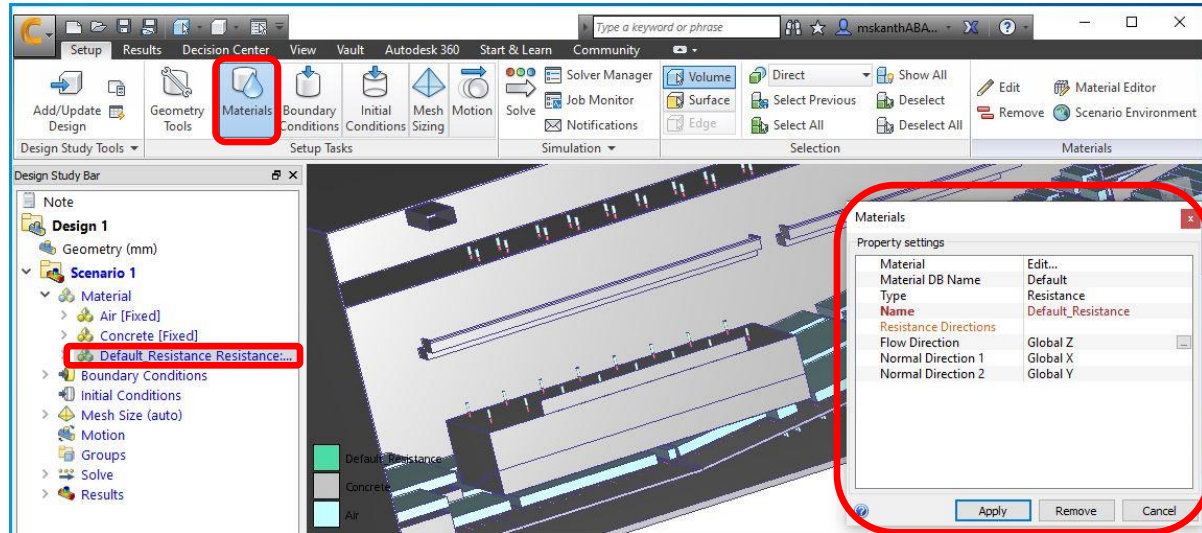


# Materials

The screenshot displays the Autodesk Inventor software interface. The top ribbon is set to the **Materials** tab, which is highlighted with a red box. The ribbon includes sections for **Design Study Tools**, **Setup Tasks**, **Simulation**, **Selection**, and **Materials**. The **Materials** section contains icons for **Edit**, **Material Editor**, **Remove**, and **Scenario Environment**. The **Design Study Bar** on the left shows a tree view for **Design 1**, with **Scenario 1** expanded to show **Material** assignments: **Air [Fixed]**, **Concrete [Fixed]**, and **Default\_Resistance Resistance...**. The main 3D view shows a mechanical assembly with a legend at the bottom left indicating material colors: green for **Default\_Resistance**, grey for **Concrete**, and light blue for **Air**. A scale bar at the bottom indicates dimensions in millimeters (0, 25440.3, 50880.7, 76321). A coordinate system (x, y, z) is visible in the bottom right corner.

# Materials

- The default resistance is provided to give resistance to the flow with a free area ratio of 0.5
- The structures are assigned as concrete material
- The fluid volume filling the room will be air material
- Humans are assigned as air volume with heat generation capacity mentioned in boundary condition



# Boundary Conditions (BCs)- Outlets

The screenshot displays the Autodesk Inventor software interface. The top ribbon is set to the 'Vault' tab, where the 'Boundary Conditions' icon is highlighted with a red box. Below the ribbon, the 'Design Study Bar' on the left shows a tree view for 'Design 1' > 'Scenario 1' > 'Boundary Conditions'. The list includes numerous 'Total Heat Generation' entries and a 'Pressure(0 Pa Gage)' entry at the bottom, which is also highlighted with a red box. A red arrow points from this entry to a small red rectangular feature on the 3D model of a mechanical part. A legend at the bottom left of the 3D view identifies the boundary condition types: Volume Flow Rate (light blue), Pressure (orange), Temperature (teal), Slip/Symmetry (blue), and Total Heat Generation (brown). The 'Pressure' entry in the legend is also highlighted with a red box. The 3D model shows a grey rectangular component with a red rectangular feature on its top surface, which is the location of the pressure boundary condition.

# BCs - Inlets

30 CFM through each  
supply diffusers

The screenshot displays a CAD software interface for a Design Study. On the left, the Design Study Bar shows a tree view with the following structure:

- Note
- Design 1
  - Geometry (mm)
  - Scenario 1
    - Material
      - Air [Fixed]
      - Concrete [Fixed]
      - Default\_Resistance Resistan...
    - Boundary Conditions
      - [Total Heat Generation(539...)]
      - [Total Heat Generation(728...)]
      - [Total Heat Generation(674 ...)]
      - [Total Heat Generation(741...)]
      - [Total Heat Generation(822...)]
      - [Total Heat Generation(788...)]
      - [Total Heat Generation(405 ...)]
      - [Total Heat Generation(944 ...)]
      - [Total Heat Generation(849...)]
      - [Total Heat Generation(755...)]
      - [Total Heat Generation(943...)]
      - [Total Heat Generation(262...)]
      - [Total Heat Generation(471...)]
      - [Total Heat Generation(337...)]
      - [Total Heat Generation(103...)]
      - [Total Heat Generation(424...)]
      - [Total Heat Generation(472 ...)]
      - [Total Heat Generation(519...)]
      - [Total Heat Generation(151...)]
      - [Total Heat Generation(741...)]
      - [Total Heat Generation(364...)]
      - [Slip/Symmetry]
      - [Pressure(0 Pa Gage)]

The 3D model shows a building's HVAC system with green arrows indicating airflow through supply diffusers. A red arrow points from the Boundary Conditions dialog box to the diffusers. The dialog box is titled "Boundary Conditions" and has the following settings:

Property settings	
Type	Volume Flow Rate
Unit	ft3/min
Time	Steady State
Volume Flow Rate	30
Direction	Reverse Normal
Fully Developed	<input type="checkbox"/>

Below the dialog box is a legend with the following items:

- Volume Flow Rate (highlighted with a red box)
- Pressure
- Temperature
- Slip/Symmetry
- Total Heat Generation

# BCs – Heat Loads

Total heat load was distributed to each air volume indicating occupants proportionate to the area.

Design Study Bar

Note

Design 1

- Geometry (mm)
- Scenario 1
  - Material
    - Air [Fixed]
    - Concrete [Fixed]
    - Default\_Resistance Resistan...
  - Boundary Conditions
    - [Total Heat Generation(539...
    - [Total Heat Generation(728...
    - [Total Heat Generation(674 ...
    - [Total Heat Generation(741 ...
    - [Total Heat Generation(822...
    - [Total Heat Generation(788...
    - [Total Heat Generation(405 ...
    - [Total Heat Generation(944 ...
    - [Total Heat Generation(849...
    - [Total Heat Generation(755...
    - [Total Heat Generation(943...
    - [Total Heat Generation(262...
    - [Total Heat Generation(471...
    - [Total Heat Generation(337...
    - [Total Heat Generation(103...
    - [Total Heat Generation(424...
    - [Total Heat Generation(472 ...
    - [Total Heat Generation(519...
    - [Total Heat Generation(151)...
    - [Total Heat Generation(741)...
    - [Total Heat Generation(364...
    - [Volume Flow Rate(30 ft3/...
    - [Slip/Symmetry]
    - [Pressure(0 Pa Gage)]
  - Initial Conditions

Legend:

- Volume Flow Rate
- Pressure
- Temperature
- Slip/Symmetry
- Total Heat Generation

Boundary Conditions

Property settings

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

Output Bar

On Plane 2- Location(X=153386,Y=22589.7,Z=-5651.93)- Value 20.5024 Celsius

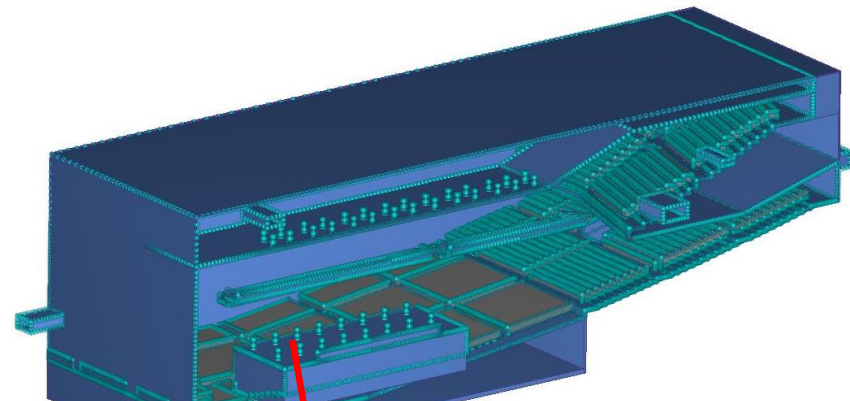
5 Volume(s) selected

# Automatic Meshing

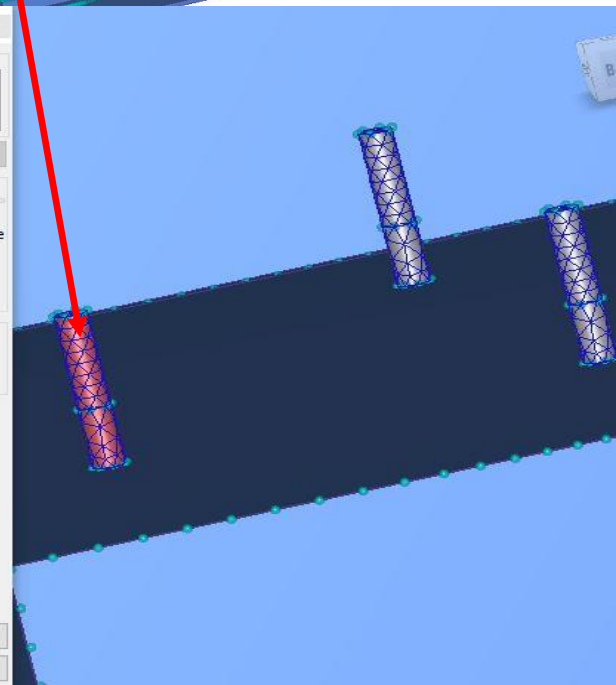
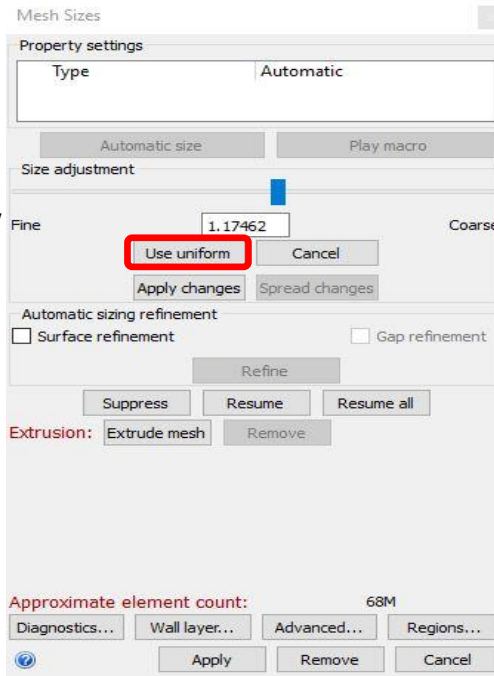
The image shows the Autodesk Inventor software interface. The top ribbon includes tabs for Setup, Results, Decision Center, View, Vault, Autodesk 360, Start & Learn, and Community. The Mesh Sizing tool is highlighted in the ribbon. The Design Study Bar on the left shows a tree view for Design 1, including Scenario 1 with sub-items like Material, Boundary Conditions, Mesh Size (auto), Motion, Groups, Solve, and Results. The central 3D model displays a mechanical part with a blue mesh applied to its surfaces. The Mesh Sizing dialog box is open on the right, showing the following settings:

- Property settings:** Type is set to Automatic.
- Buttons:** Automatic size, Play macro, Use uniform, Cancel, Apply changes, Spread changes, Refine, Suppress, Resume, Resume all, Extrude mesh, Remove.
- Size adjustment:** A slider between Fine and Coarse.
- Automatic sizing refinement:** Surface refinement and Gap refinement are both unchecked.
- Approximate element count:** 68M.
- Bottom buttons:** Diagnostics..., Wall layer..., Advanced..., Regions..., Apply, Remove, Cancel.

# Uniform Meshing

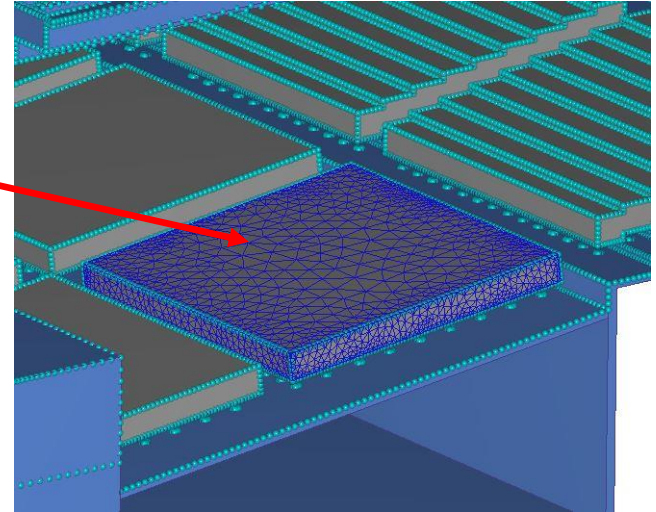


Uniform fine mesh (at least 4-5 elements) on diffusers to ensure proper air flow through the supply diffusers



# Uniform Meshing

Uniform mesh (at least 4-5 elements) to capture effective heat absorption in air volume for occupants



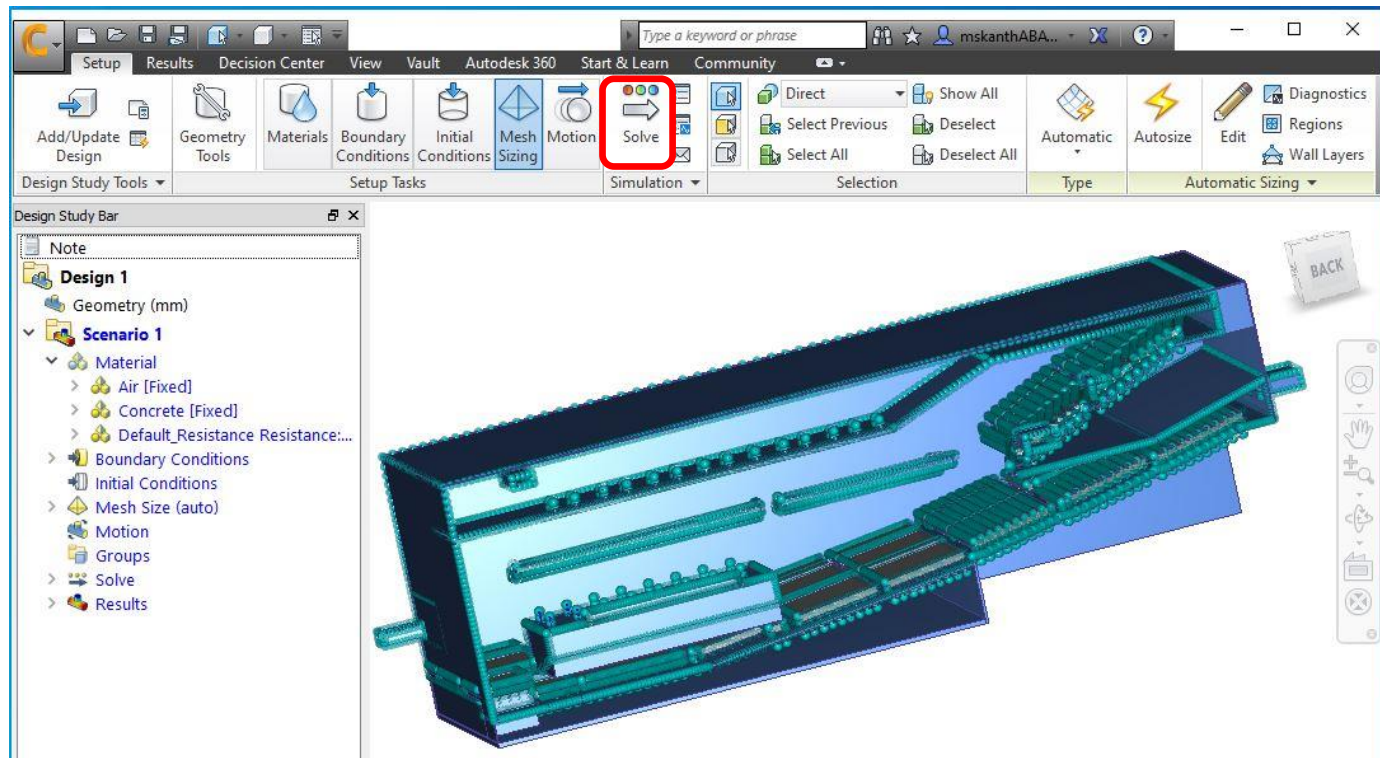


# **Set up Simulation and Visualize Results**

# Simulation and results

## Autodesk CFD Solve and Results Workflow

- Solver control
- Physics (Flow and Heat Transfer)
- Results – Global / Plane
- Results – Airflow Velocity and Temperature

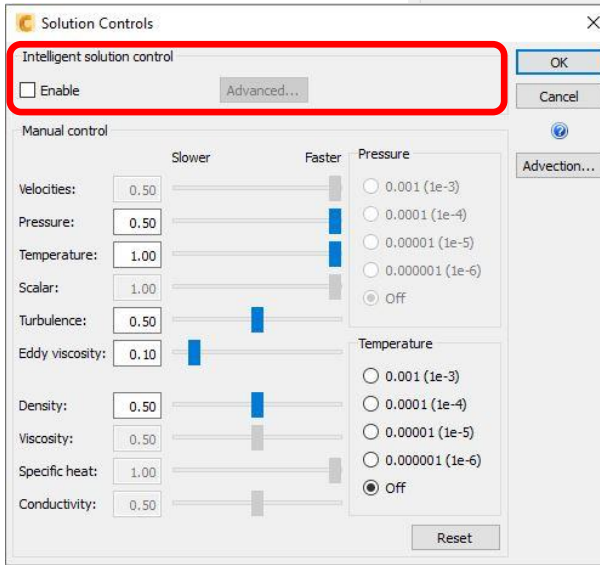
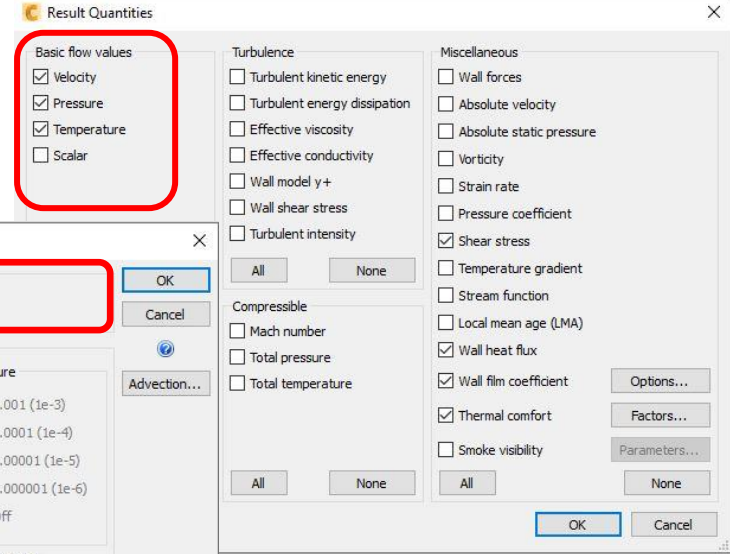
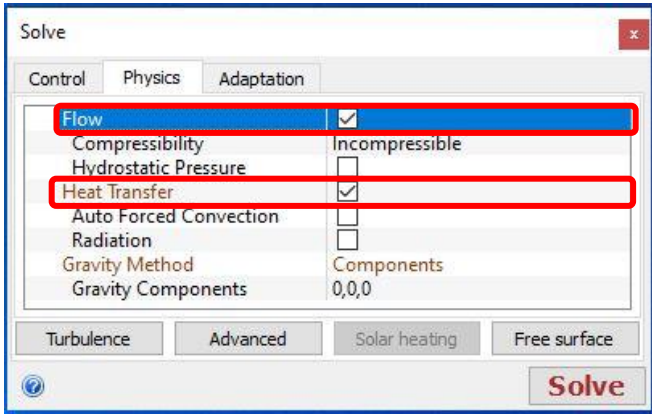
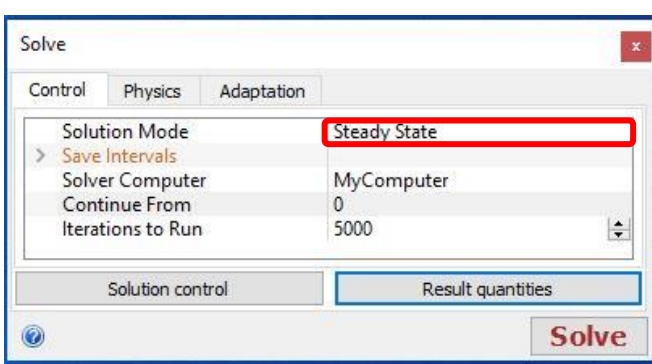


# Solve

The image shows the Autodesk Inventor software interface. The ribbon is set to the 'Simulation' tab, and the 'Solve' button is highlighted with a red box. The 'Design Study Bar' on the left shows a tree view with 'Design 1' expanded to 'Scenario 1', which includes 'Material', 'Boundary Conditions', 'Initial Conditions', 'Mesh Size (auto)', 'Motion', 'Groups', 'Solve', and 'Results'. The main 3D view shows a mechanical assembly with a mesh. A 'Solve' dialog box is open, also highlighted with a red border, showing the following settings:

Control	Physics	Adaptation
Solution Mode	Steady State	
> Save Intervals		
Solver Computer	MyComputer	
Continue From	0	
Iterations to Run	5000	

Buttons for 'Solution control' and 'Result quantities' are visible at the bottom of the dialog box, along with a 'Solve' button.



# Solver settings

Steady state solution, Intelligent Solution Control, Flow and Heat Transfer, Forced Convection (Gravity method unselected)

# Results- Global

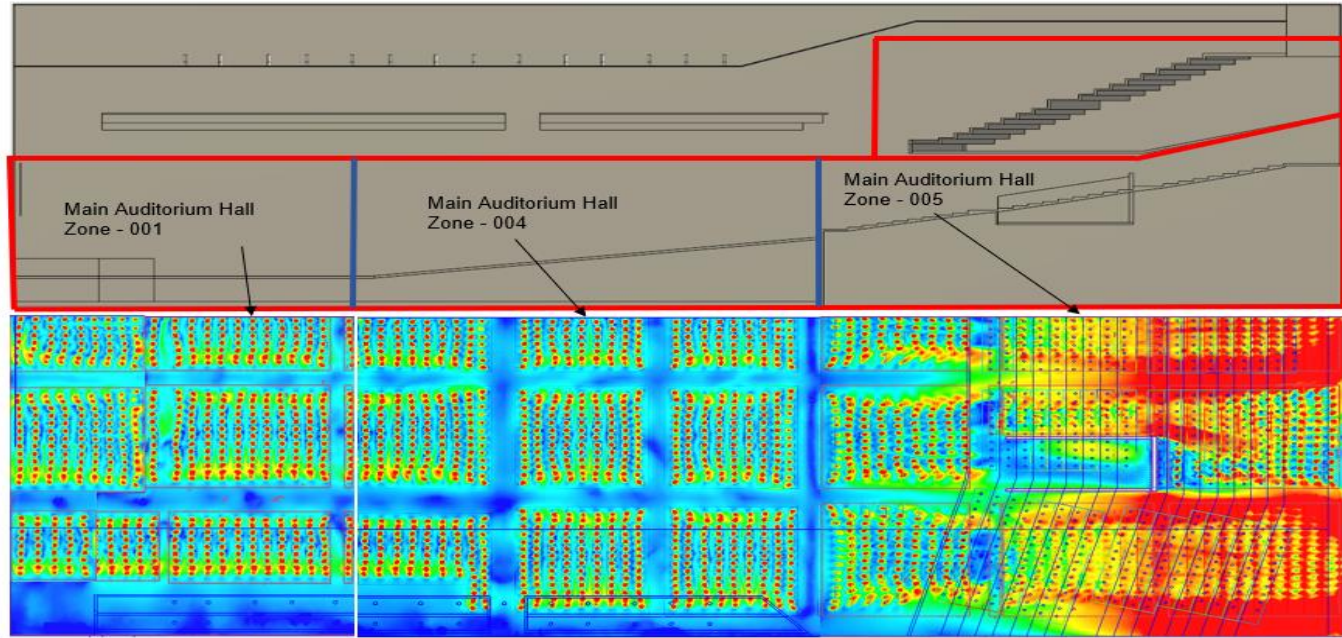
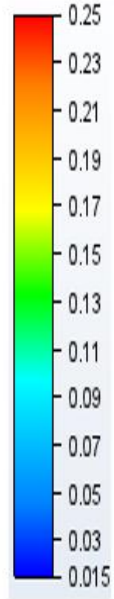
The screenshot displays the Autodesk Inventor interface with the **Results** tab selected. The ribbon includes various visualization tools such as **Global** (highlighted with a red box), **Planes**, **Traces**, **Iso Surfaces**, **Iso Volumes**, **Wall Calculator**, **Parts**, and **Points**. The **Results Tasks** section contains **Report Generator**. The **Review** section includes **Status File**, **Summary File**, and **Setup File**. The **Iteration/Step** dropdown is set to **1210 Last**. The **Global Result** is set to **Temperature**, and the **Global Vector** is set to **None**. The **Design Study Bar** on the left shows a tree view for **Design 1** with **Scenario 1** expanded, listing **Material** (Air, Concrete, Default\_Resistance), **Boundary Conditions**, **Initial Conditions**, **Mesh Size (auto)**, **Motion**, **Groups**, **Solve**, and **Results**. The main workspace shows a 3D model of a mechanical part with a temperature simulation. A color scale on the left indicates temperature in Celsius, ranging from 14 (blue) to 24 (red). The part is predominantly yellow and green, with a red section at the bottom. A **BACK** button is visible in the top right corner of the 3D view, and a vertical toolbar with navigation icons is on the right side.

# Results- Plane

The screenshot displays the Autodesk Inventor interface with the following components:

- Top Ribbon:** Includes tabs for Setup, Results, Decision Center, View, Vault, Autodesk 360, Start & Learn, and Community. The Results tab is active, showing tools like Global, Planes (highlighted with a red box), Traces, Iso Surfaces, Iso Volumes, Wall Calculator, Parts, and Points. Reporting tools include Report Generator, Status File, Summary File, and Setup File. Iteration/Step controls show '1210 Last' and 'Solve'.
- Design Study Bar:** Shows the active study '(6) Temperature - Celsius'. The left pane lists the design tree: Design 1, Geometry (mm), Scenario 1, Material (Air [Fixed], Concrete [Fixed], Default\_Resistance\_Resistance:..., Boundary Conditions, Initial Conditions, Mesh Size (auto), Motion, Groups, Solve, Results).
- Simulation View:** A 3D model of a mechanical part is shown with a temperature distribution. A color scale on the left ranges from 14 (blue) to 24 (red). The model shows a high-temperature region (red) on the bottom surface, transitioning through yellow and green to a lower temperature region (blue) on the top surface. A blue arrow points to a specific plane within the model.
- Right Panel:** Contains a 'Planes' dropdown menu with 'Temperature' selected for the Result and 'None' for the Vector. Other options include Bulk and XY Plot. A 'Make Summary' button is also present.

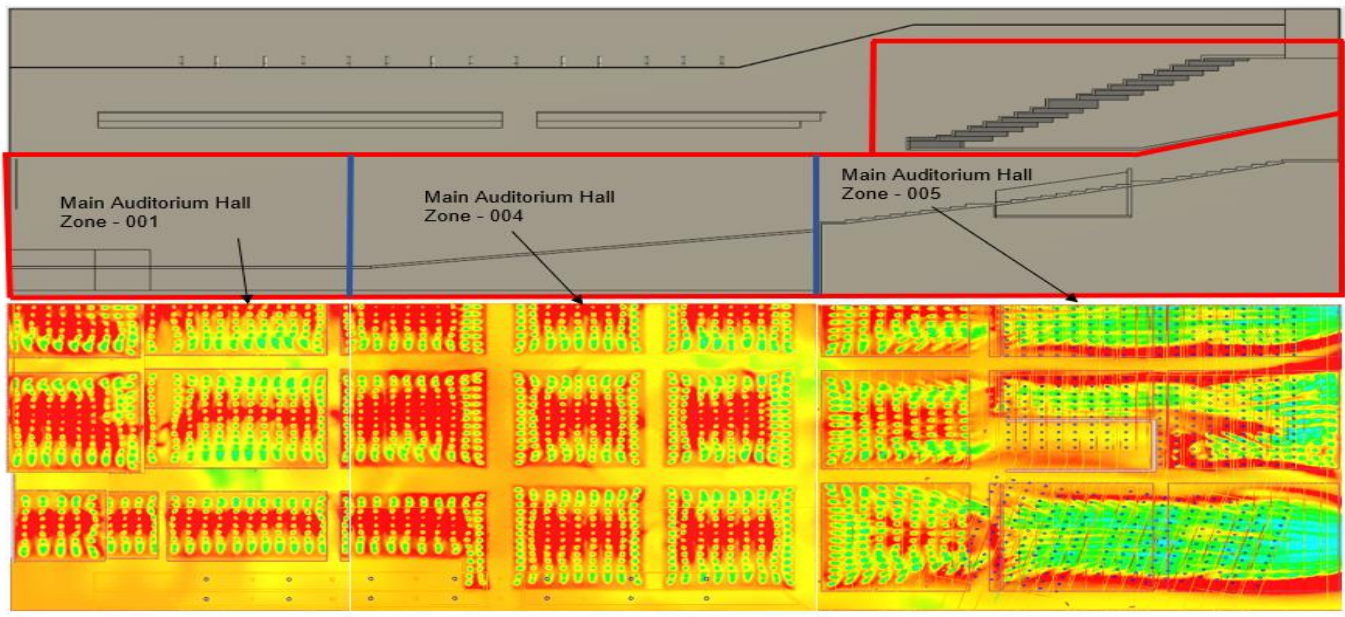
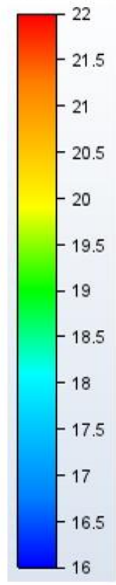
Velocity m/s



## Main Auditorium Hall -Results- Velocity at 1.7 m level

Results on Cut Planes are used to visualize velocity data on 3D model. Plan view shows high velocity through each diffusers indicated by red dots.

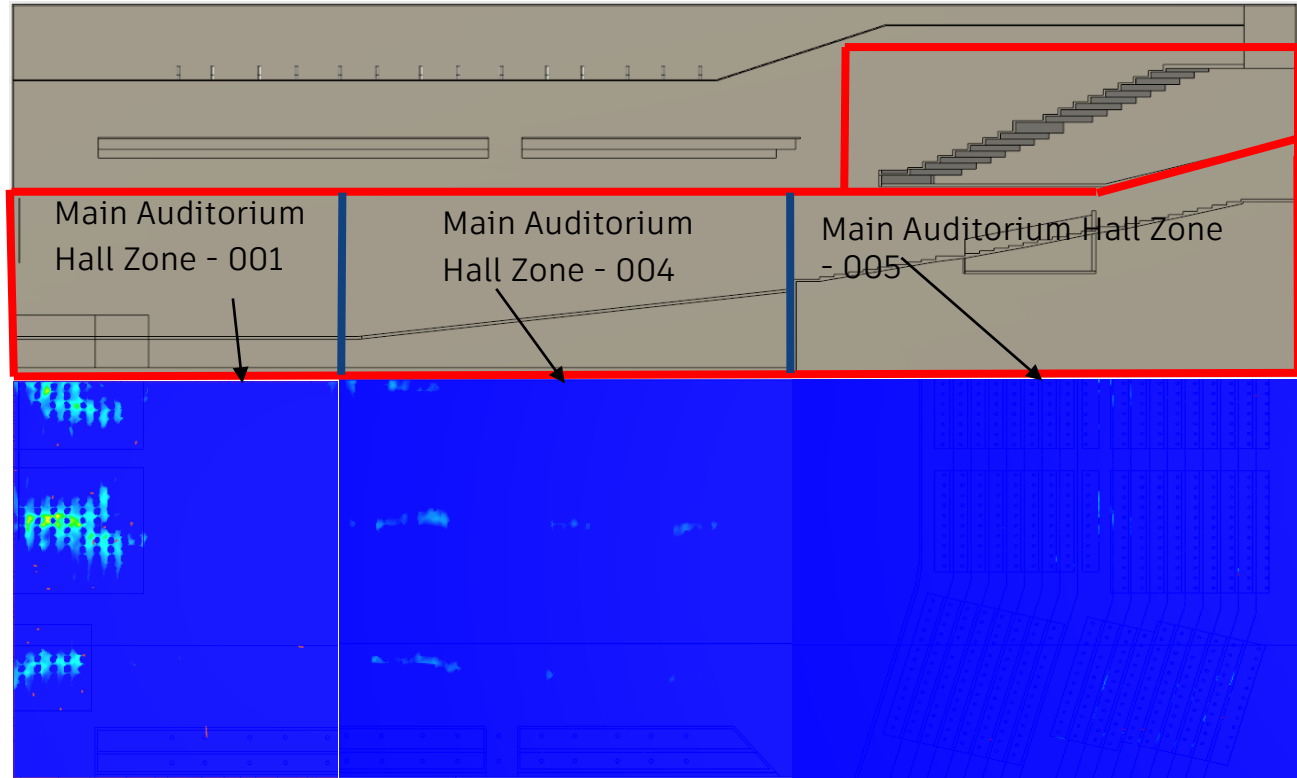
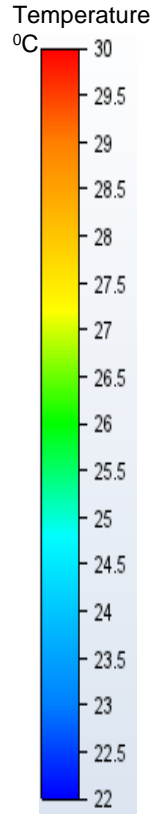
Temperature °C



# Main Auditorium Hall - Results- Temperature at 1.7 m level

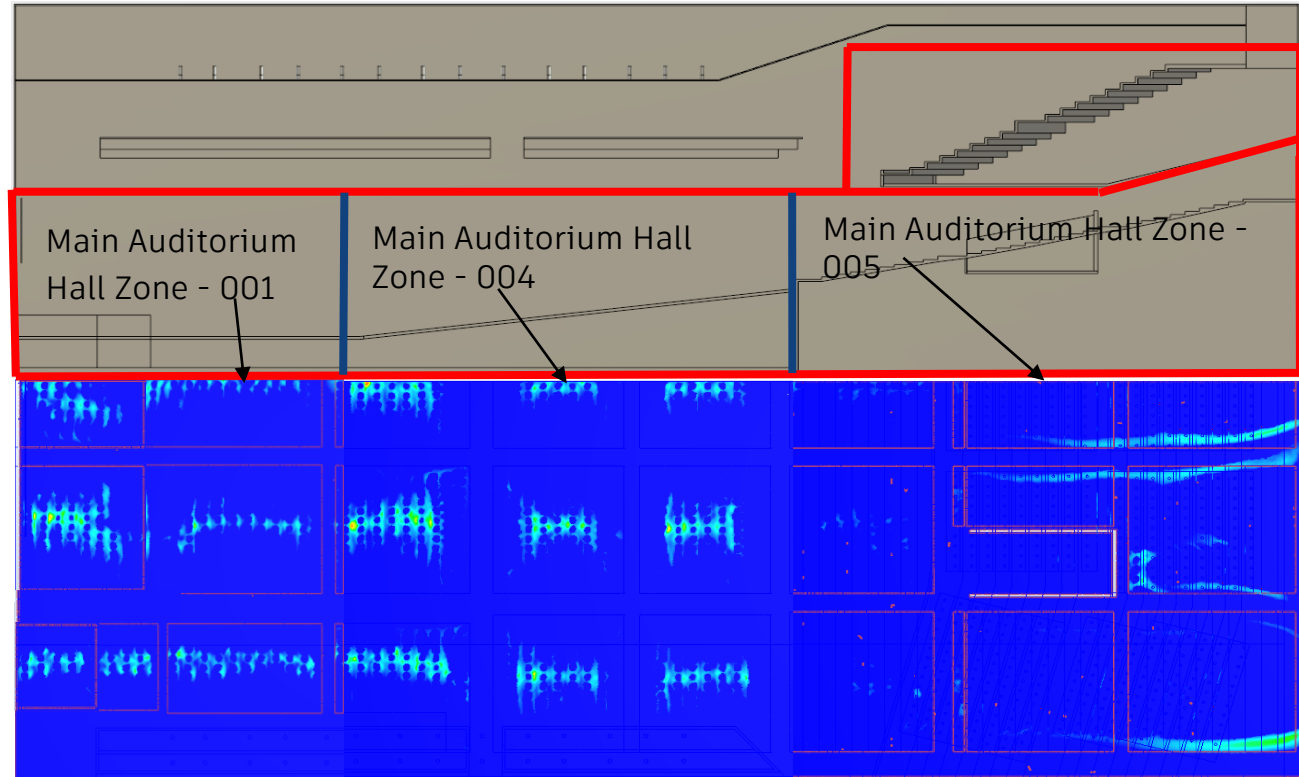
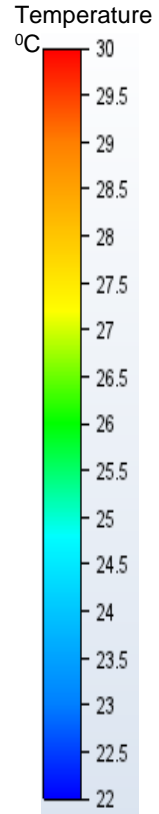
Results using Cut Plane at 1.7m level from FFL can be used to evaluate temperature for personnel comfort. In this case , 22<sup>0</sup> C.

# Main Auditorium Hall - Results- Temperature at 1.7 m level



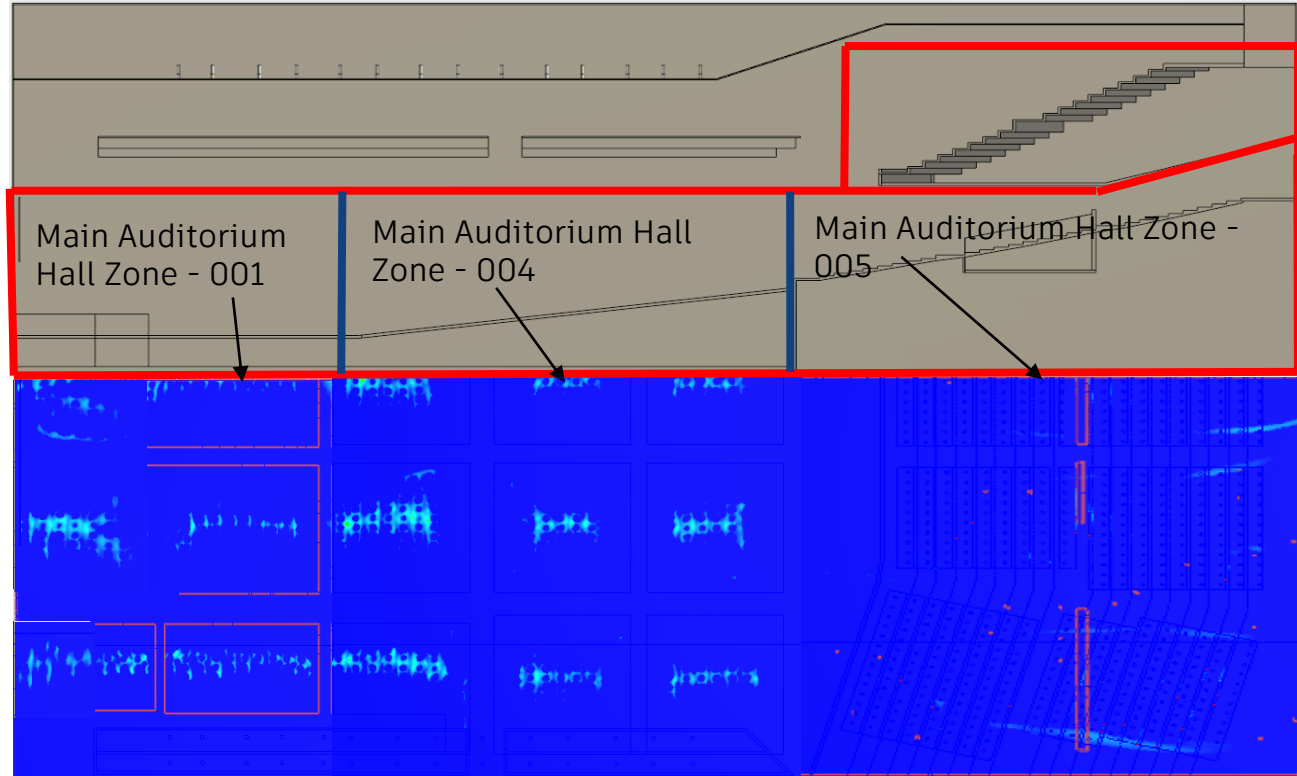
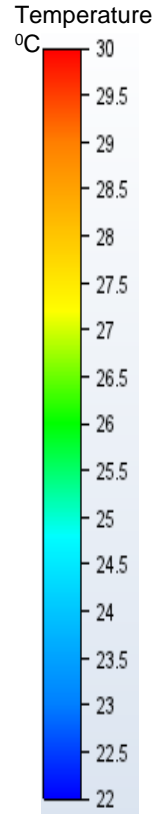
Temperature contour at 1.7 m from the floor

# Main Auditorium Hall - Results- Temperature at 0.7 m level



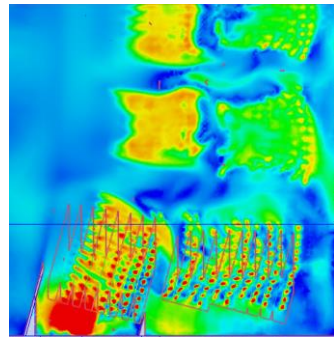
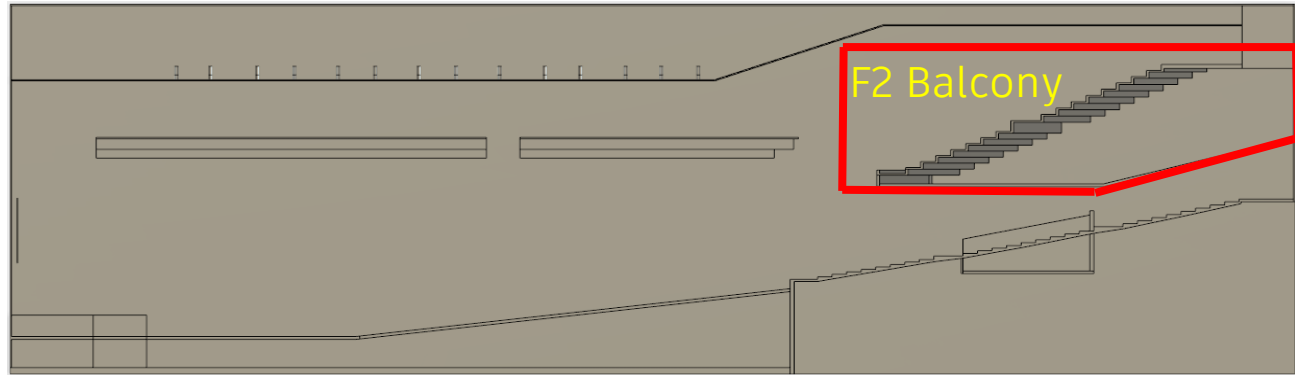
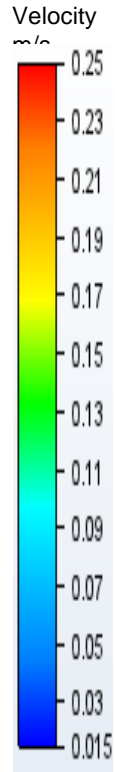
Temperature contour at 0.7 m from the floor

# Main Auditorium Hall - Results- Temperature at 1.1 m level

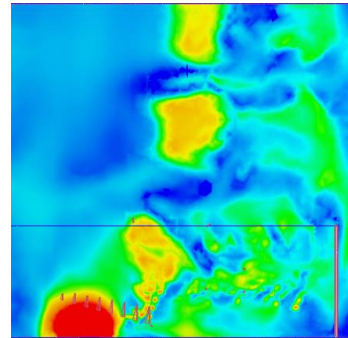


Temperature contour at 1.1 m from the floor

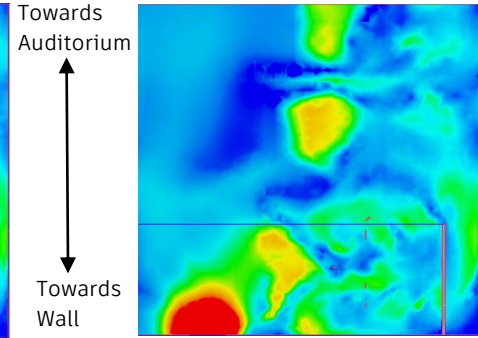
# F2 Balcony CFD analysis results (Air flow pattern)



Velocity contour at  
0.7 m from the floor



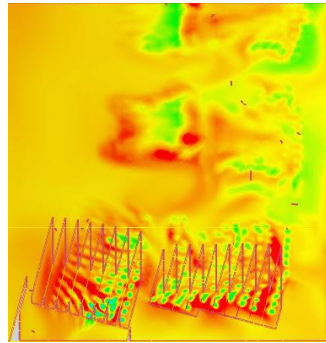
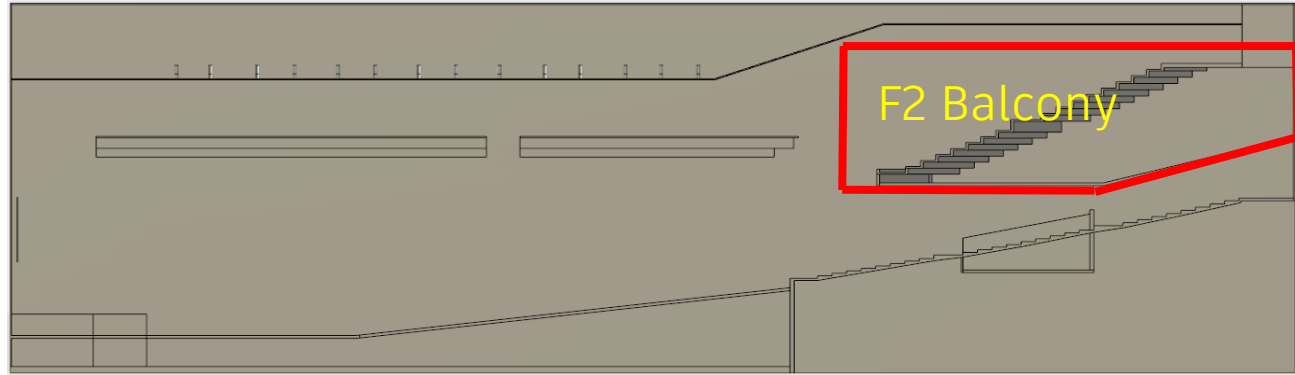
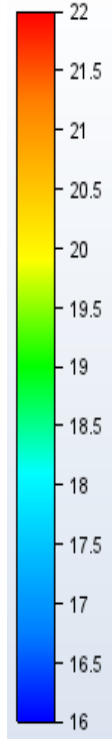
Velocity contour at  
1.1 m from the floor



Velocity contour at  
1.7 m from the floor

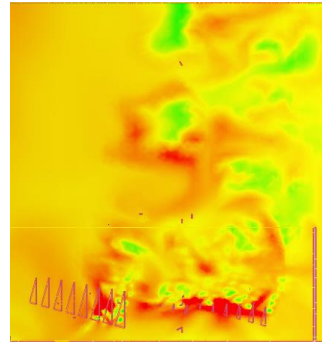
# F2 Balcony CFD analysis results (Temperature)

Temperature



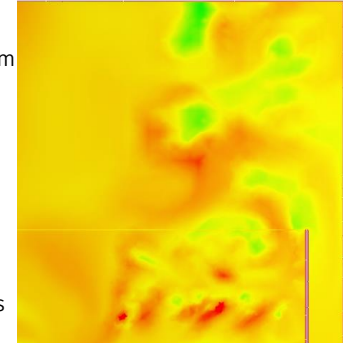
Towards Auditorium  
↑  
↓  
Towards Wall

Temperature contour at 0.7 m from the floor



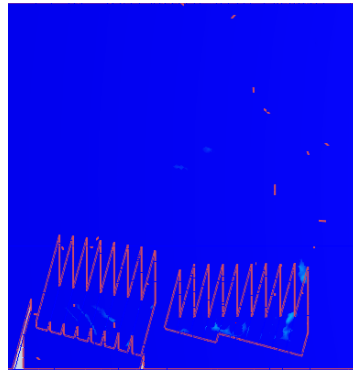
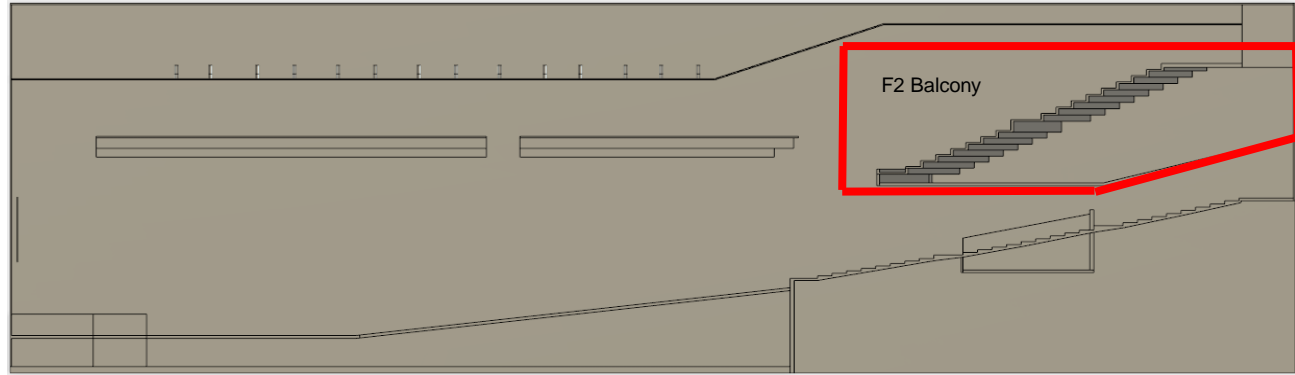
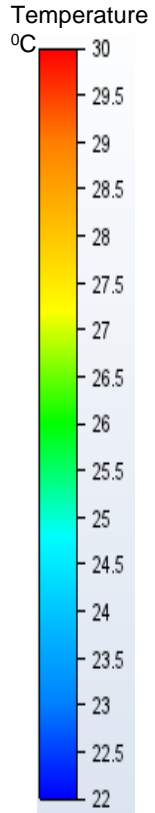
Towards Auditorium  
↑  
↓  
Towards Wall

Temperature contour at 1.1 m from the floor

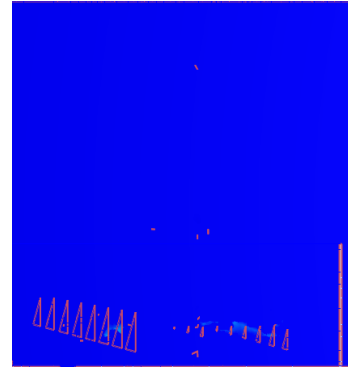


Temperature contour at 1.7 m from the floor

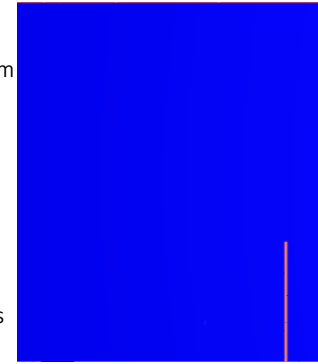
# F2 Balcony CFD analysis results (Temperature)



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

**Review results and  
evaluate design**

# Auditorium Hall- Thermal Comfort Evaluation

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 within the recommended value of less than 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 is adequate.

# Wrap up

## Key learnings:

- Identify specific workflows in Autodesk CFD
- Implement workflows
- Assess CFD results
- Validate design for thermal comfort in a large auditorium hall

***Thank you for your attention and happy AU2021!***

The background features a dark, almost black, space with several large, metallic, angular shapes that resemble parts of a mechanical assembly or a futuristic interface. These shapes are rendered with soft highlights and shadows, giving them a three-dimensional appearance. The central focus is the text 'AUTODESK UNIVERSITY' in a bold, white, sans-serif font.

# AUTODESK UNIVERSITY

Autodesk and the Autodesk logo are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product offerings, specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document.

© 2021 Autodesk. All rights reserved.