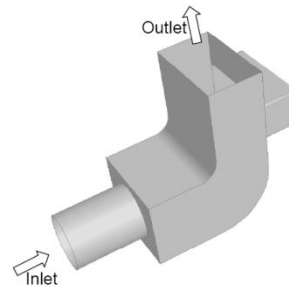
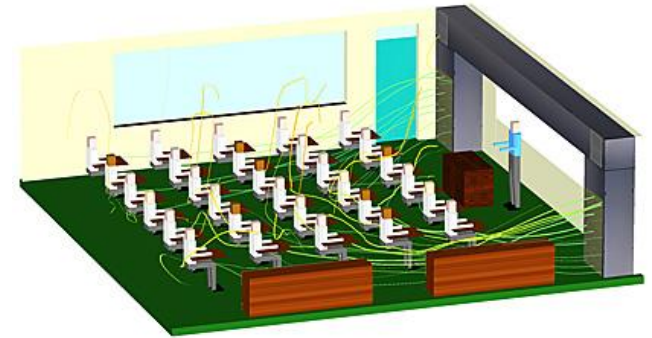


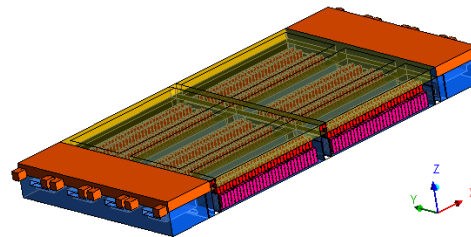
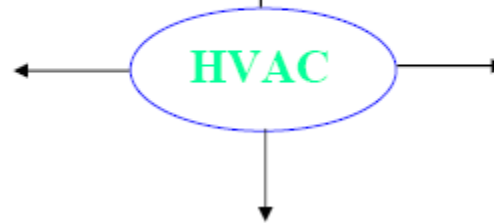
Aero & Auto interior HVAC design for human comfort and safety



Design and optimization of HVAC ducts



HVAC design of offices to attaining max. human comfort with least power consumption



Design of High performance data centers for uniform cooling of servers



## Case Studies

1. Design optimization of ducts by aero acoustic performance.
2. CFD Analysis of Data Centre Cooling
3. HVAC Analysis for Basement of Multistoried building

## Design optimization of ducts by aero acoustic performance.

### Objectives :

- CFD analysis to predict noise sources.
- Total acoustic power emitted for all designs.
- To predict Predict aero-noise “loudness” rankings with Broadband Noise Source Models.
- To optimise the design.

### Boundary Condition

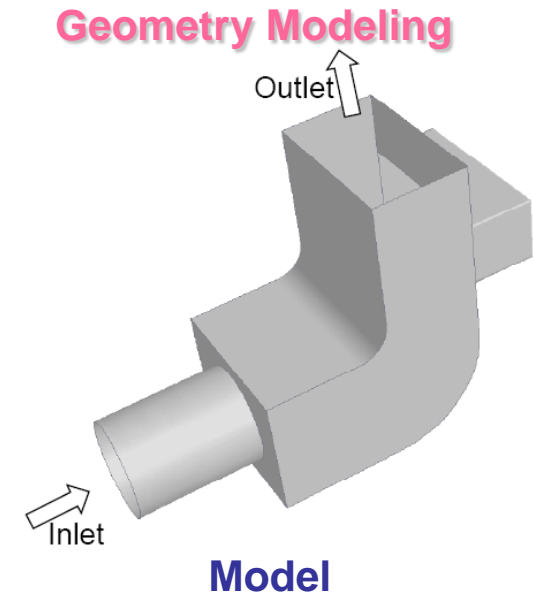
Inlet : Mass Flow Rate

Outlet : Static Pressure

Acoustic model

### Methodology & Convergence

- The flow is Incompressible
- Solver used in the analysis is FLUENT.
- RNG k- $\epsilon$  Turbulence Model and Broadband noise source models are considered .
- The convergence criteria from FLUENT is taken where the scaled residuals decrease to  $10e-4$  for all equations.
- Boundary of the fluid domain is usually a no-slip wall which would constrain fluid velocity at walls to zero.

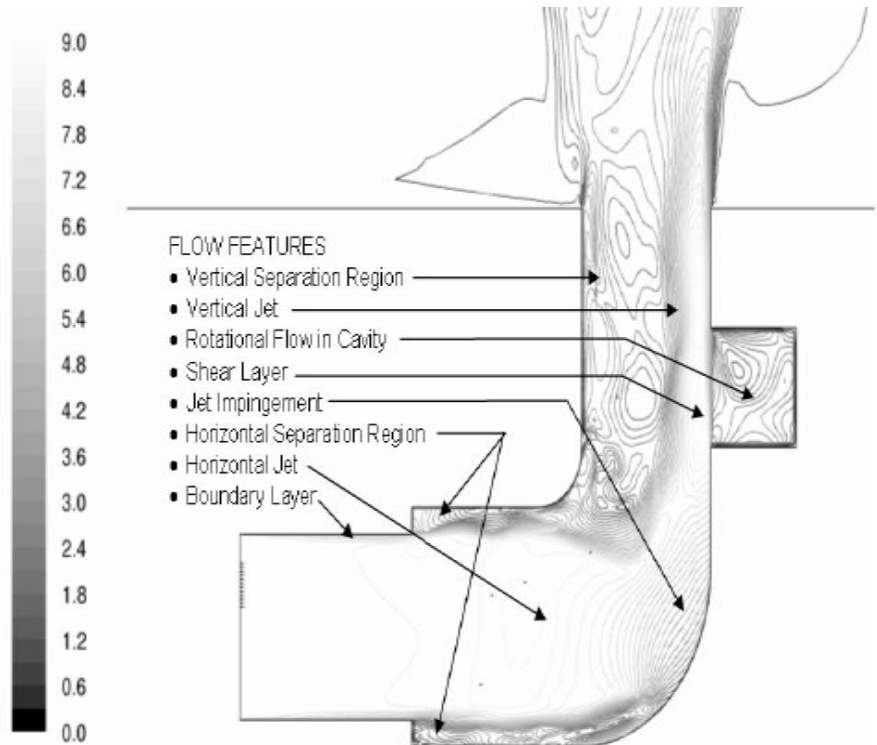


## Task Executed:

The model is created in Unigraphics. Hexahedral meshing is carried out using ICM-Hexa. Analysis is carried out in FLUENT after applying the material properties and different boundary conditions. Different plots are provided in the technical report using FLUENT.

## OBSERVATIONS:

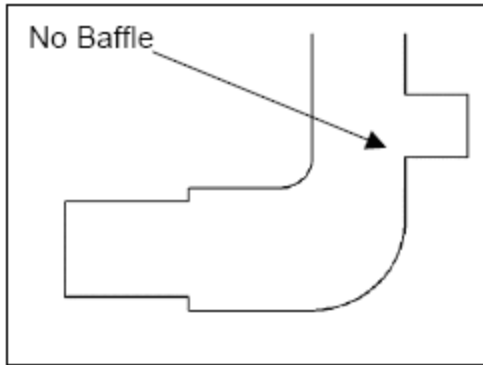
- Base line design was analyzed
- Acoustic source has been identified
- Baffle addition will reduce the noise level intensity.
- Designs were tested with different sizes of the baffle



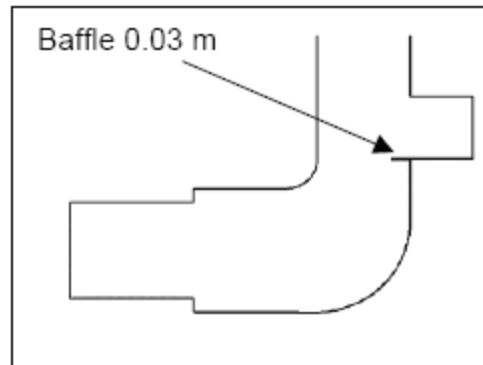
Contours of velocity magnitude (m/s)



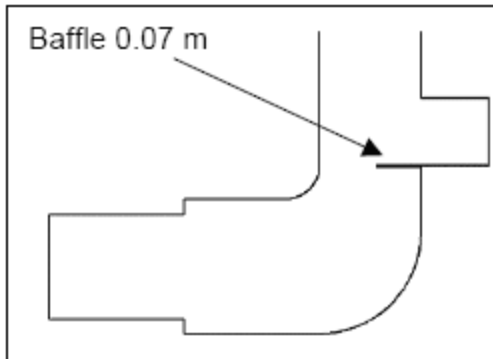
## Design Variations



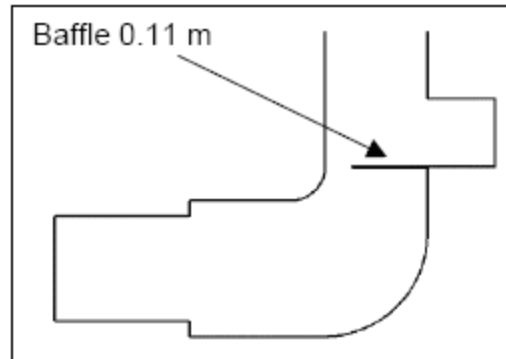
Baseline: Design 1



Design 2



Design 3



Design 4

Design	Noise Source Power (W) Volume Integral of Pressure
Design 1	7.31e-13
Design 2	1.63e-12
Design 3	9.48e-12
Design 4	2.94e-10

Simulated duct loudness rankings

Design4 > Design3 >  
Design2 > Design1



## Summary:

Design 4 is loudest, Design 1 is the least

Aero-noise is noise caused by the flow

Noise sources are time-varying stresses in the flow Field

- Transient pressure fluctuations close to the boundary create dipole sources

## Broadband Noise Source Models

- Simple tool for providing design direction
- Require only steady state flow solution
- Inexpensive compared to transient solution

## HVAC Analysis for Basement of Multistoried building

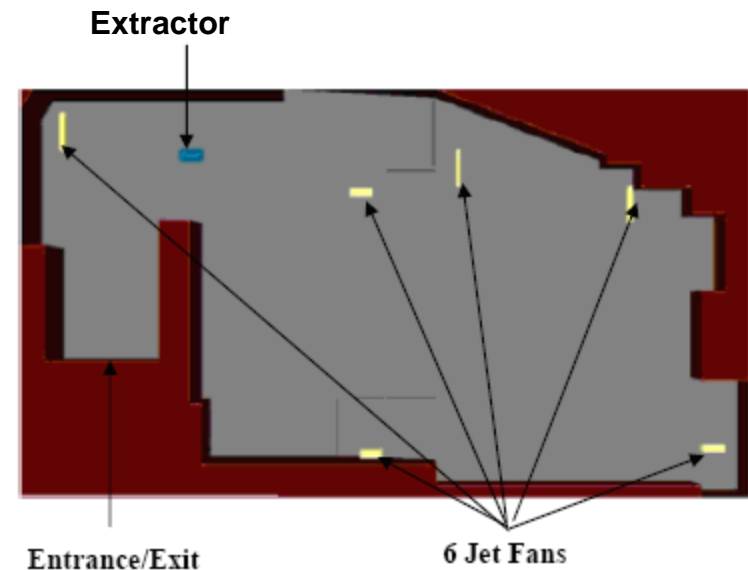
### Objectives :

- Propagation of fire in car parking area
- Extent of fire propagation with respect to time
- Fan locations to dissipate the smoke incase of fire

### Geometry Modelling & Meshing



**Site Location**



**Model**



## Analysis Methodology

### a. Boundary Condition

Inlet BC for Jet Fans (6): Mass flow rate & Total Temperature

Outlet BC for Extractor (1): Static Pressure

Fire : Fire has created by using methane.

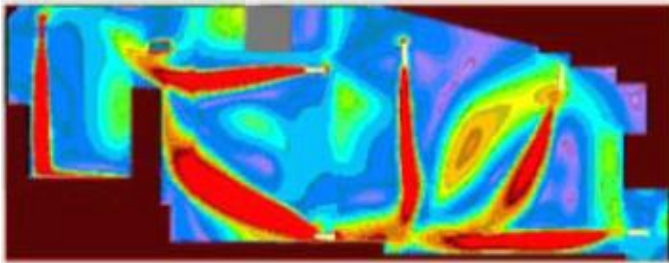
### b. Methodology & Convergence

- The flow is Incompressible
- Solver used in the analysis is CFX.
- k- $\epsilon$  Turbulence Model is considered with Standard log law wall function near wall treatment with Inlet turbulence Intensity= 5%.
- The convergence criteria from CFX is taken where the scaled residuals decrease to  $10e-4$  for all equations.
- Multi species combustion and reaction was modeled.
- Boundary of the fluid domain is usually a no-slip wall which would constrain fluid velocity at walls to zero.

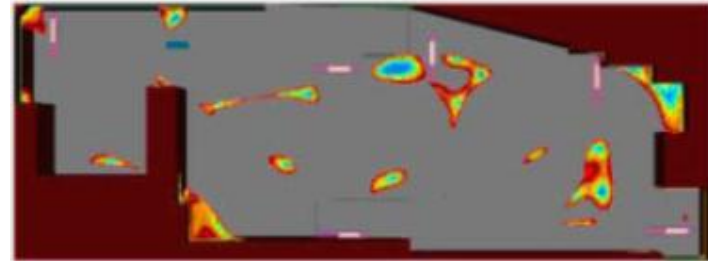
### Task Executed:

The model is created in Unigraphics. Hexahedral meshing is carried out using ICEM-Hexa. Analysis is carried out in CFX after applying the material properties and different boundary conditions. Different plots are provided in the technical report using CFX- Post-processing.

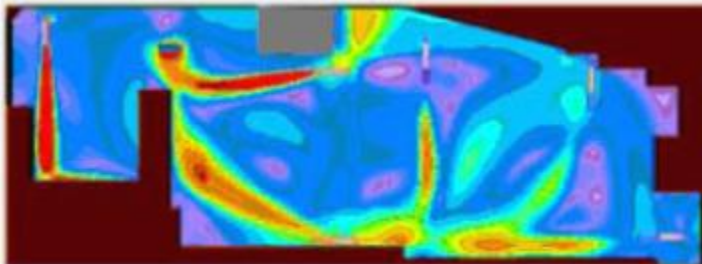
## Result & Discussion



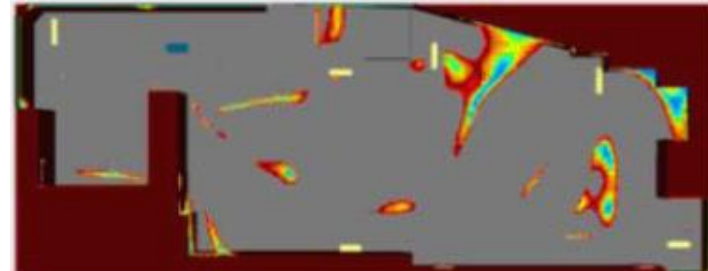
Velocity Pattern at 1.8m (adult head height)



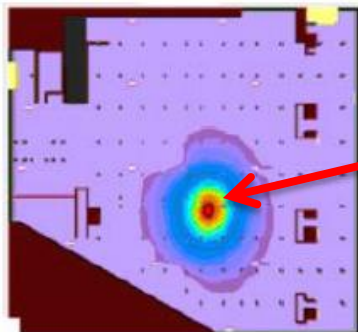
Air pockets below critical air speed (1.8 m)



Velocity Pattern at 0.5m (child head height)

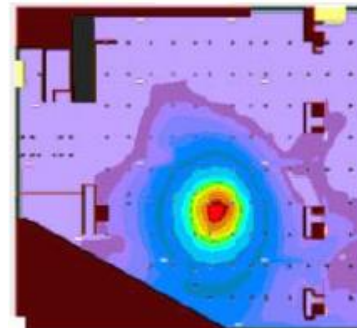


Air pockets below critical air speed (0.5 m)



**Fire origin**

First 2min of fire propagation



Fire spread after 8min



## Final Results :

- Speed plots are provided to show the speed of the air in basement at adult head height (1.8 m) and child head height (0.5 m)
- From the Analysis, the static pressure, total pressure, temperature, velocity contours and velocity vectors at different location are plotted.
- Based on the analysis, locations of air speed less than critical speed has optimized.
- Fire propagation has been predicted.

## Conclusion :

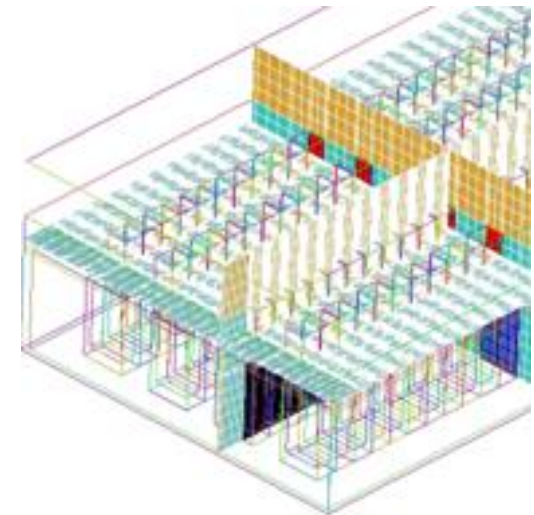
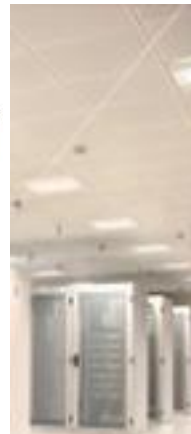
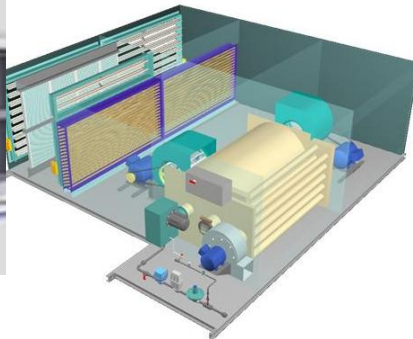
- Small localized regions are permitted as long as there is sufficient air speed in majority of the area is above the permitted limits
- Customer optimized flow circulation based on CFD analysis for different boundary conditions both in steady state and transient.
- The CFD analysis help in reduction in design time and cost involved in real time experimental for different conditions.
  
- Evacuation time and procedure to be followed has identified when fire occurs
  
- Fan locations to dissipate the smoke incase of fire

## CFD Analysis of Data Centre Cooling

### Objectives :

- To perform a 3D CFD analysis to predict the airflow/ temperature profile/pattern for different operating condition
- Analysis for Steady state & Transient condition
- To predict the velocity, temperature & static pressure distribution at different location.
- To optimise the design.

### Geometry Modelling & Meshing



**Model**

**Mesh**



## Analysis Methodology

### a. Boundary Condition

Inlet BC: Mass flow rate & Total Temperature

Outlet BC: Static Pressure

AHU Inlet Fan & Outlet Fan : Fan characteristics applied

Cabinet : Dissipation power rate and the fan characteristics applied

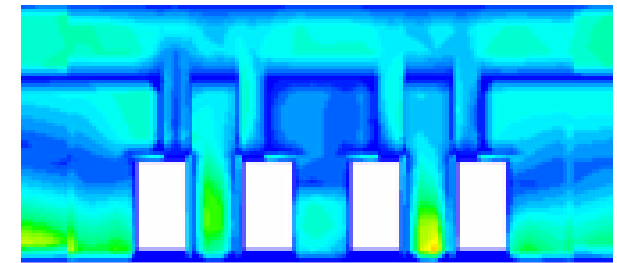
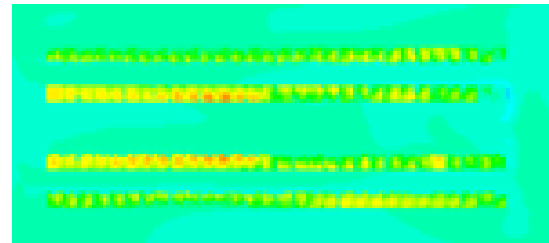
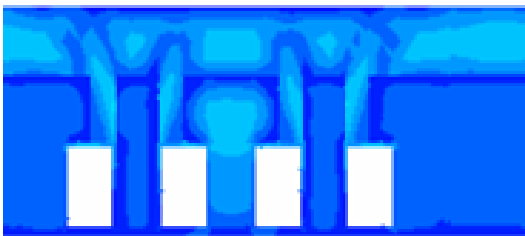
### b. Methodology & Convergence

- The flow is Incompressible
- Solver used in the analysis is CFX.
- k- $\epsilon$  Turbulence Model is considered with Standard log law wall function near wall treatment with Inlet turbulence Intensity= 5%.
- The convergence criteria from CFX is taken where the scaled residuals decrease to  $10e-4$  for all equations.
- Heat dissipation will be taken care of by material properties and flow status at the solid-fluid interface.
- Boundary of the fluid domain is usually a no-slip wall which would constrain fluid velocity at walls to zero.

## Result & Discussion

### Task Executed:

The model is created in Unigraphics. Hexahedral meshing is carried out using ICEM-Hexa. Analysis is carried out in CFX after applying the material properties and different boundary conditions. Different plots are provided in the technical report using CFX- Post-processing.



Typical contour plots

### Final Results :

From the Analysis, the static pressure, total pressure, temperature, velocity contours and velocity vectors at different location are plotted. Based on the analysis, the hot areas and recirculation zones are predicted and design is optimized to meet the required design performance.

### Conclusion :

Customer optimized the data centre design based on CFD analysis for different boundary conditions both in steady state and transient. The CFD analysis help in reduction in design time and cost involved in real time experimental for different load conditions.