THERMAL AND AIRFLOW SIMULATION OF THE GULBENKIAN GREAT HALL

Nuno Mateus[1], Guilherme Carrilho da Graca[1]
[1]University of Lisbon, Lisbon, Portugal

ABSTRACT

This paper presents an application of building thermal and CFD simulation in the refurbishment of the HVAC system of the Gulbenkian great hall in Lisbon (Portugal). The existing overhead mixing ventilation system was converted into displacement ventilation.

The thermal simulation tool EnergyPlus was used to size the HVAC system. CFD simulations where used to predict detailed airflow velocity and temperatures in the space and to assess the precision of the mixed and displacement ventilation models that are currently available in EnergyPlus.

INTRODUCTION

The Gulbenkian great hall in Lisbon is one of the most important engineering and architectonic building projects of the 20th century in Portugal. Built in 1960's, it has a seating capacity of 1100 on the main floor plus 200 seats on a balcony area with up to 250 artists in the stage area (see figure 1).

The hall has four main areas: stage, orchestra pit, stalls and balcony. The existing HVAC system uses a single air-handling unit (AHU) that serves all the areas through an overhead mixing system. After forty years of intense use, the existing HVAC system doesn't meet current requirements for indoor air quality and thermal comfort. The main problems are excessive energy use during rehearsal periods, inefficient mixing of the ventilation system.


Audience view
Stage view

Figure 1 – Gulbenkian great hall.

After four decades of continuous use there is a need for a complete refurbishment of the HVAC and lighting systems. This need created an opportunity to update the systems, but also to change the airflow pattern from overhead mixing to displacement ventilation (DV). The transition between systems is achieved by transforming the existing air exhaust openings, located under the seats, into inflow openings, allowing the use of a displacement system (the current standard approach in contemporary concert hall design). Figure 2 shows the existing ventilation strategy.

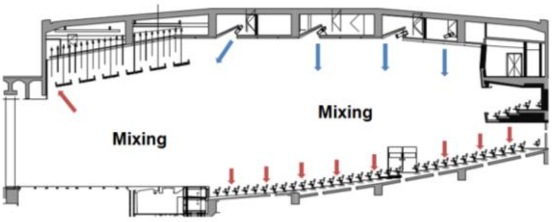

Figure 2 – Gulbenkian great hall existing HVAC system.

For the new system, the coexistence of different indoor thermal comfort requirements in the room dictated the use of three distinct air-handling units:

- Seating area (stalls and balcony).
- Stage area.
- Orchestra pit.

The geometry of the orchestra pit makes the use of an overhead system difficult (there is no ceiling), clearly, in this case, displacement ventilation is the better option. In contrast, the functional requirements of the stage area make the use of a displacement system with low-level inflow diffusers, extremely difficult. For this reason, a variable configuration system is used for the stage. The proposed HVAC system configuration for the refurbishment is shown in figure 3. In its initial configuration the system included a potentially problematic interaction between an overhead system (stage system with high momentum nozzles) and a displacement ventilation system (seating area, with low velocity inflow). The HVAC system was sized using the thermal building simulation software EnergyPlus (DOE-LBNL, 2010). The complexity of the proposed system and the uncertainty about the interaction of the two different airflow patterns created the need for a computational fluid dynamics simulation (CFD), focusing on the main occupancy and internal load scenarios.

THERMAL SIMULATION

In order to analyse indoor air conditions of each occupied space, the simulation model includes four thermal zones (stage, orchestra pit, stalls and balcony, shown in figure 3). The hall has heavy concrete construction. The floor and walls are made of 0.3m concrete (0.27 W/m.K; 750 kg/m^3; 1000 J/kg.K), the roof consists of 0.3m concrete slab (2 W/m.K; 2100 kg/m^3; 880 J/kg.K) with external insulation (0.04 W/m.K; 0.02m) and a ventilated air cavity.

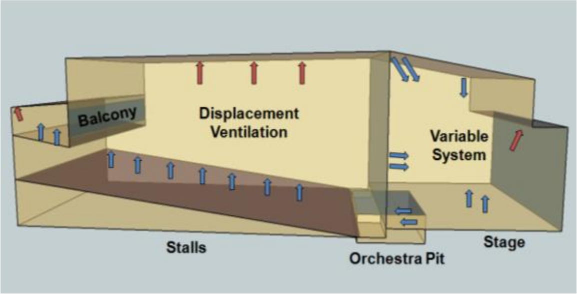

Figure 3- Great hall thermal zones.

The proposed HVAC system, composed of three air-handling units, was modelled using EnergyPlus template for a unitary system. Table 2 shows the settings used for each air-handling unit. The stalls and balcony thermal zones were modelled using the detailed three node UCSD displacement ventilation model that is available in EnergyPlus (Carrilho da Graça, 2003, 2004). This model reproduces a thermal air stratification, with three distinct zones: floor zone (lowest temperature); occupied zone (temperature felt by the occupants); and upper zone (highest temperature). The other zones (orchestra pit and stage) were treated as perfectly mixed.

Internal loads scenarios

Two typical maximum stage utilization scenarios were defined to size the system, labelled Classical and Modern (see table 1). Both scenarios used the same occupancy schedule, corresponding to the maximum, fully occupied daily utilization (from 10 am to 12 am; from 3 pm to 5 pm; from 7 pm to 9; from 9.30 pm to 11.30 pm).

Sizing criteria

The thermal building simulation sizing results are dependent on the weather file used. Since 2009, the Typical Meteorological Year (TMY) became the new standard weather file used in energy based simulations (Gates, 2011). The weather files used in EnergyPlus simulations were created based on this format through statistical methods, and typically, the extreme weather data were smoothed by average values.

Typically, a TMY weather file for Lisbon (EnergyPlus Weather) would be used to run the simulations. Due to the great sensibility of this project, a weather data sensibility analysis was performed to select adequate weather data to size the HVAC system.

Two additional sources of sizing weather data were considered: TMY weather data, eight years measured weather data (airport of Lisbon). A third weather data source was created, based on airport weather data and ASHRAE sizing conditions (ASHRAE Handbook Fundamentals, 2009).

Enthalpy values were analysed and a 0.4 and 1 percentile, heating (minimum enthalpy) and cooling design day (maximum enthalpy), for the three sources of weather data were defined.

Table 1 - Sizing criteria –loads considered in different scenarios.

Zone	Scenario	Occupancy	Internal Gains		
			Occupants (W/occupant)	Illumination (W/m^2)	Total (W/m^2)
Stalls	Classical music	1091	104	25	175
	Modern	1091	104	25	175
Balcony	Classical music	163	104	10	150
	Modern	163	104	10	150
Stage	Classical music	200	167	166	310
	Modern	40	167	292	367
Orchestra Pit	Classical music	0	-	-	-
	Modern	60	167	10	206

Table 2 – UHA’s sizing criteria.

UHA	Setpoints		Maximum airflow (m^3/ occupant*h)	Maximum airflow (m^3/h)
	Temperature (ºC)	HR (%)		
Stalls and Balcony	20 - 24	35 - 65	35	45000
Stage	21 - 23	40 - 60	35	10000
Orchestra Pit	21 - 23	40 - 60	40	4000

HVAC system sizing results

Figures 4-5 show results of a weather data sensibility analysis on HVAC system sizing results. Figure 4 presents sizing results for the three sources of analysed weather data.

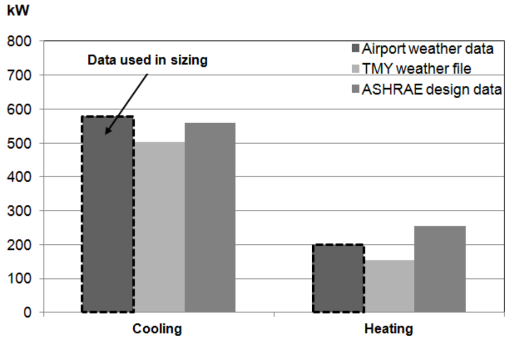

Figure 4 - Sizing results: Airport weather data, TMY weather file and ASHRAE design days sizing comparison.

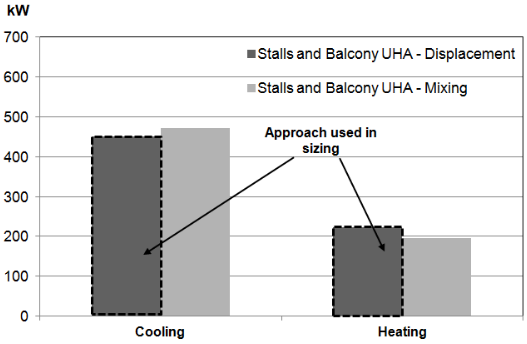

Figure 5- Results: Stalls and balcony UHA sizing - 0.4% Airport weather data.

The figure 4 shows that the definition of the HVAC system design day has a large impact on the system size. Figure 5 compares sizing results for stalls and balcony UHA, if the airflow pattern of the system is correctly modelled (displacement model) or if mixing model was used. These results demonstrate the influence of modelling properly the airflow pattern on EnergyPlus in order to rightly size the HVAC system.

The difference between the two extreme cases, for cooling and heating, shown in figure 4 is: 15% (cooling) and 65% (heating). The sizing effects of the ventilation model are lower: 4% (cooling) and 14% (heating).

CDF SIMULATION

In the present case CFD was used to confirm the EnergyPlus sizing results, assess thermal comfort (including average airflow velocity), identify the optimal inflow strategy for the stage, and test the compatibility between overhead (stage) an displacement (seating area) systems.

One of the first examples of the use of CFD to analyze the ventilation efficiency and thermal comfort of overhead mixing systems in two large auditoriums was presented by Hangan et al. in 2001. This study focused on the typical difficulties of overhead mixing systems: preventing overdraft and avoiding still air zones with poor ventilation efficiency. In a more recent applied research paper by Kavgic et al. (2008) CFD is used in a post occupancy assessment of a theatre with a poorly performing natural ventilation system. The usual pitfalls of this ventilation strategy were apparent: excessive inflow velocity (the measured inflow rate per occupant was 50% higher than the value used in the sizing presented in the next sections of this paper) and user complaints of cold ankles. A recent application of CFD in concert hall refurbishment design was presented by Scanlon et al. (2011). As in the present case, the proposed solution involved inverting existing inflow and exhaust openings in order to convert the existing overhead system into a displacement system. In this case CFD is used both to fine tune the design and to facilitate information flow within the design team.

Geometry

The geometry used in the simulation is shown in Figure 6. The CFD simulations were performed using PHOENICS (2012). Due to its robustness and adequate capabilities for room ventilation flows, the k-ε turbulence model was used in this study (with the Yap correction for confined spaces). Buoyancy was modelled using the Boussinesq approximation.

CFD simulations are time consuming processes that must be streamlined by introducing simplifications that allow for a converged simulation in adequate time (ideally less than a day for each case).

The symmetry of the auditorium along the longitudinal axis enabled for the use of a model of one-half of the room. Grouping of the audience rows was used with each modelled row representing three rows.

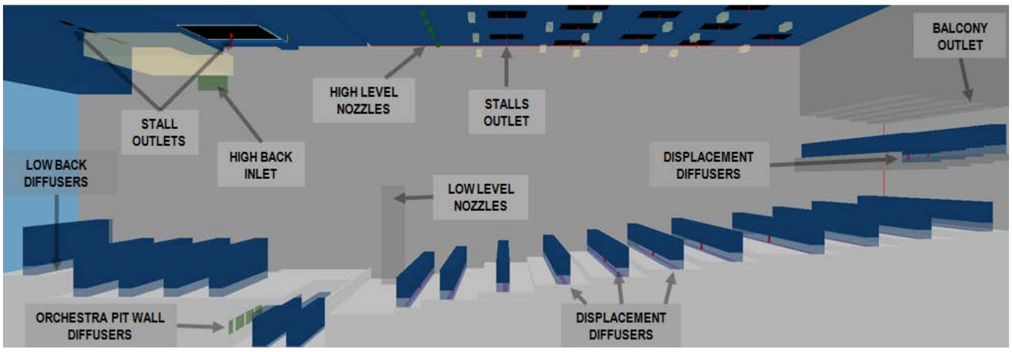

Figure 6 - Gulbenkian great hall CFD model geometry (half room).

Internal loads, like occupants and lighting were modelled as blocks with a fixed heat source composed of simulation domain material (air). The figure 7 presents the grid refinement performed along X-axis.

In order to facilitate the analysis of the results the airflow velocity and temperature were averaged in control volumes that are coincident with the thermal load volumes shown in figure 6 (in dark and light blue).

CFD simulation scenarios

Four different inflow configurations were created for the stage system (table 3). The inflow boundary conditions tested are shown in table 4.

Table 3 – CFD simulated scenarios.

Scenario name	System description
Classical HN+LN	High Nozzles (5 000m^3/h) + Low Nozzles (5 000m^3/h)
Classical LB+LN	Low Back diffusers (4 000m^3/h) + Low Nozzles (6 000m^3/h)
Modern HN+LN	High Nozzles (5 000m^3/h) + Low Nozzles (5 000m^3/h)
Modern LN+HB	Low Nozzles (6 000m^3/h) + High Back (4 000m^3/h)

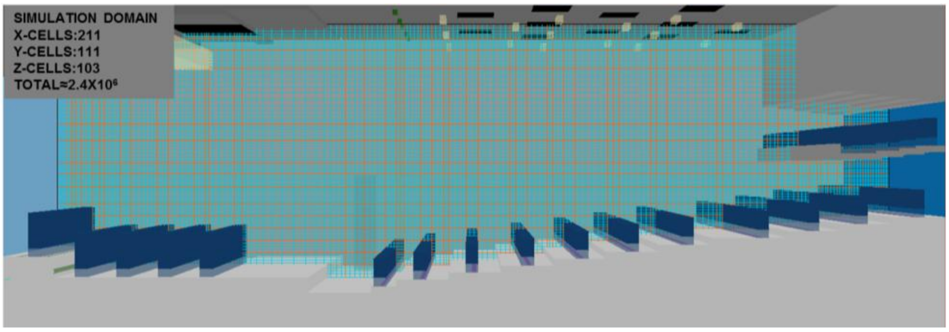

Figure 7 – Grid refinement (xx axis) of Gulbenkian great hall PHOENICS model.

Table 4 – CFD simulation conditions.

Cases	Zone	Inflow boundary conditions		Airflow volume (m^3/s)
		Velocity (m/s)	Temperature ($^\circ$C)	
All scenarios	Balcony	0.16	18	1.6
	Stalls	0.18	18	10.6
Modern scenario	Orchestra pit	0.18	19	0.7
Classical HN+LN	Stage	2.87	14	2.8
Classical LB+LN		0.13 (diffusers) 2.87 (nozzles)	14	2.8
Modern HN+LN		2.87	14	2.8
Modern LN+HB		0.13 (High back) 2.87 (nozzles)	14	2.8

CFD Results

Results of the simulated temperature and velocity in the stage, orchestra pit, balcony and stalls for the four CFD models studied are shown below (figures 8-11).

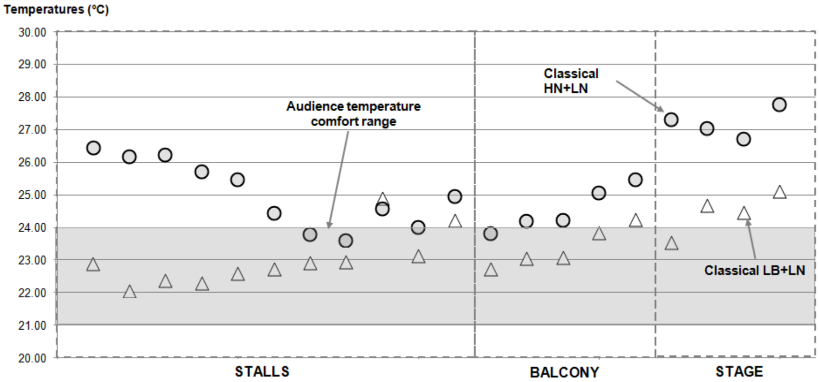

Figure 8 – Results: Room temperature Classical scenario.

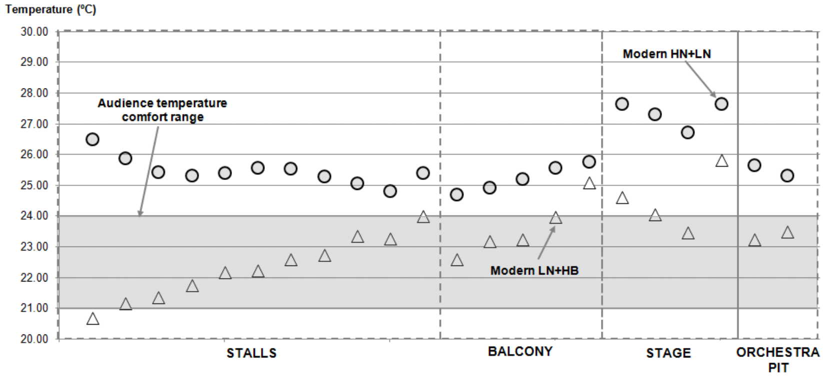

Figure 9 – Results: Room temperature Modern scenario.

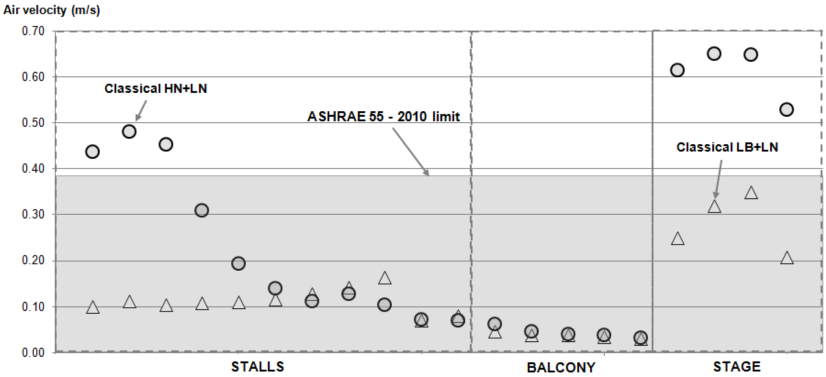

Figure 10 – Results: Room velocity Classical scenario.

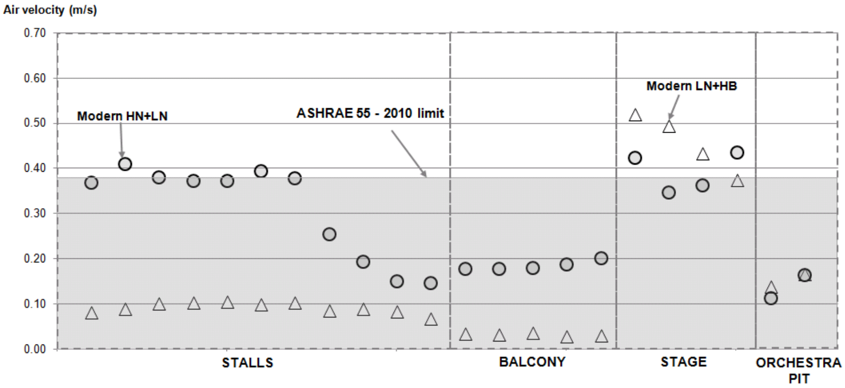

Figure 11 –Results: Room velocity Modern scenario.

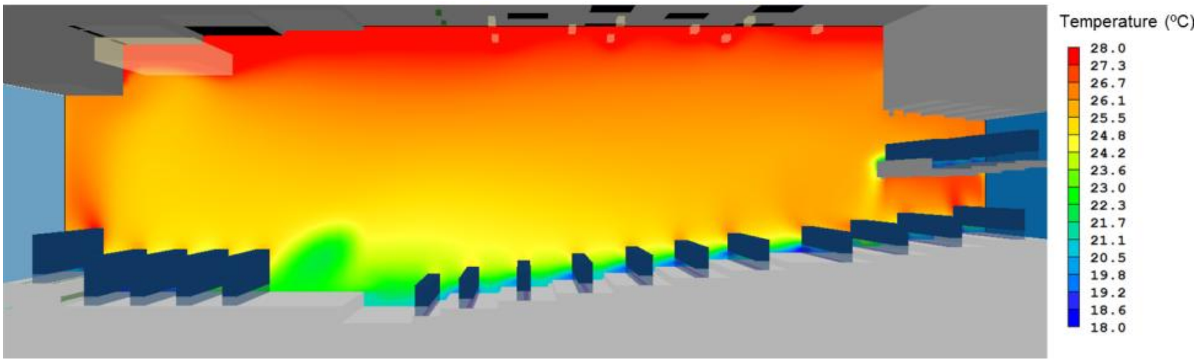

Figure 12- Results: Classical LB+LN scenario - Room temperature profile.

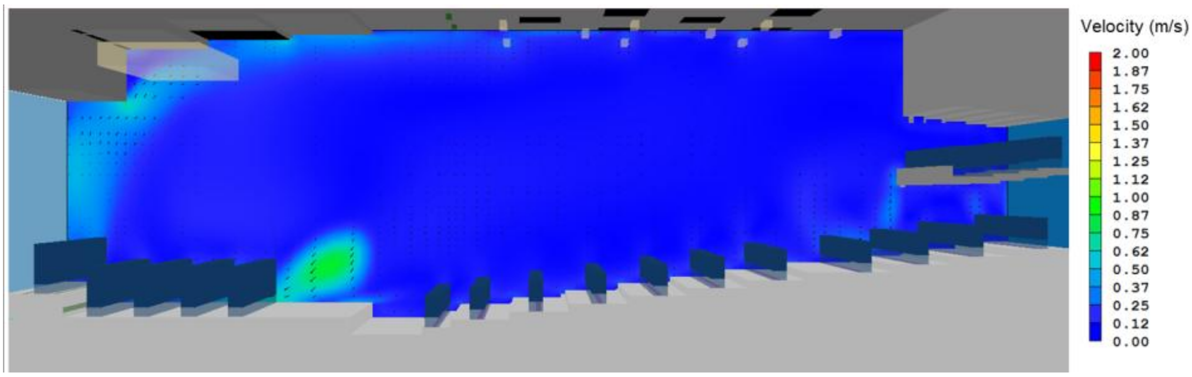

Figure 13 - Results: Classical LB+LN scenario - Room velocity profile.

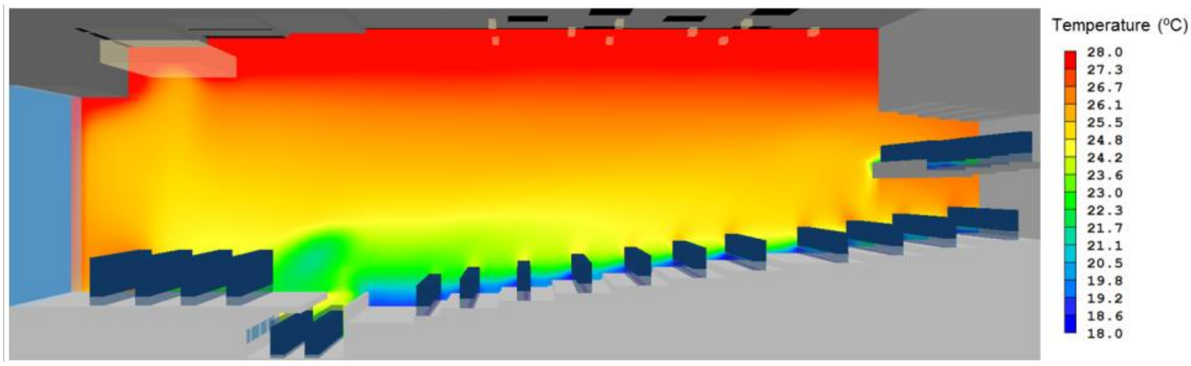

Figure 14– Results: Modern LN+HB scenario - Room temperature profile.

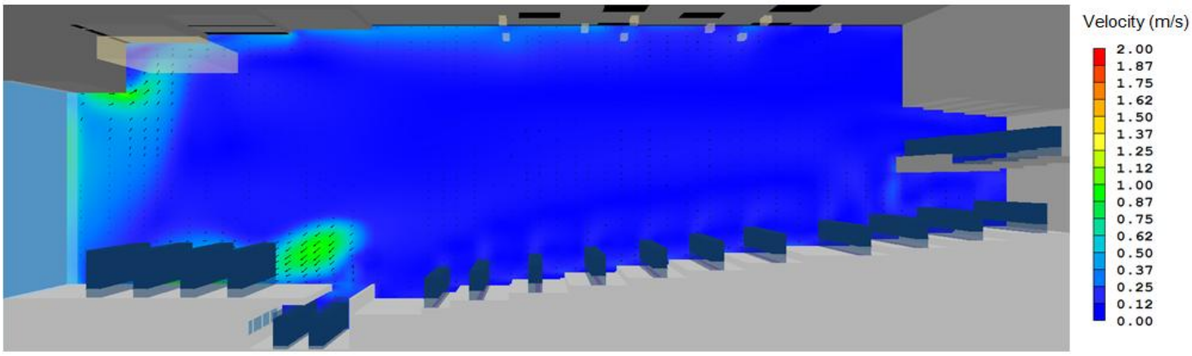

Figure 15 – Results: Modern LN+HB scenario - Room velocity profile.

Figures 8-11 show that only two of the four simulated cases (Classical LB+LN and Modern LN+HB) present temperature and air velocity values within the target requirements (shown in grey in the figure). Figures 12-15 show PHOENICS temperature and air velocity results for these two, better performing, scenarios.

Detailed analysis of the CFD results for the two scenarios (Classical HN+LN and Modern HN+LN) that did not perform as expected show that the low velocity airflow along the ceiling (in the positive x direction), caused by the large stage heat gains, greatly reduces the momentum flow from the high level nozzles (that should reach the stage). Due to this effect, the penetration depth of these jets is reduced to approximately ten diameters (2.4 meters, see figure 16). This effect is in agreement with experimental work by Andreopoulos *et al.* (1984) and Chan *et al.* (1998), showing that a high velocity jet perpendicular, or head on, to a uniform airflow with a lower velocity mixes within ten diameters of the outflow nozzle.

Figures 12-15 show that the proposed configuration works well and there is no relevant negative interaction between the displacement ventilation system used for the seating are and hybrid system used for the stage. The back row of the main seating area is affected by heat accumulation under the balcony and may benefit from additional dedicated extraction.

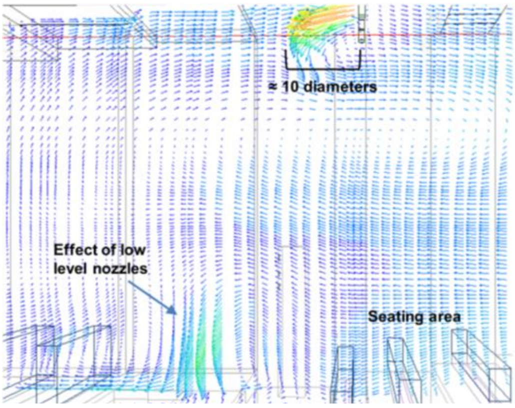

Figure 16 – Classical HN+LN and Modern HN+LN: high nozzle velocities profile showing fast velocity decay (and thereby limited cooling effect).

COMPARISON BETWEEN DETAILED AND SIMPLIFIED SIMULATIONS

This section presents a comparison between EnergyPlus and CFD simulations results for the different stage HVAC configurations scenarios (Classical LB+LN and Modern LN+HB).

Table 5 shows a predicted average temperature comparison between CFD and EnergyPlus using two distinct modeling approaches: perfectly mixed (single node) and stratified (three node UCSD displacement ventilation model).

The comparison results, presented in table 5, show good agreement in all zones, except the stage. The discrepancies can be attributed to the difficulty of EnergyPlus to model a system that does not correspond to a typical mixing or displacement configuration.

Table 5 – EnergyPlus and CFD simulations zones temperature results comparison.

Scenario	Simulation	Zone			
		Stalls	Balcony	Stage	Orchestra pit
Classical	*E+* DV	23.0	23.3	21.5	-
	E+ Mixing	25.0	25.0	25.1	-
	CFD	23.0	23.4	24.2	-
	Difference DV (°C)	**0.1**	**0.1**	**2.7**	**-**
	Difference Mixing (°C)	**-2.0**	**-1.6**	**-0.9**	
Modern	*E+* DV	23.1	23.4	-	-
	E+ Mixing	24.8	24.9	21.5	24.2
	CFD	22.3	23.6	24.5	23.4
	Difference DV (°C)	**-0.7**	**0.2**	**-**	**-**
	Difference Mixing (°C)	**-2.5**	**-1.3**	**3.0**	**-0.9**

Table 6 – EnergyPlus and CFD simulations zones temperature results comparison: Displacement model heat gains distribution fraction change (0.4 to 0.5).

Scenario	Simulation	Zone			
		Stalls	Balcony	Stage	Orchestra pit
Classical	*E+* DV	23.3	23.6	21.8	-
	CFD	23.0	23.4	24.2	-
	Difference (°C)	**-0.3**	**-0.2**	**2.4**	**-**
Modern	*E+* DV	23.4	23.7	21.4	24.2
	CFD	22.3	23.6	24.5	23.4
	Difference (°C)	**-1.1**	**-0.1**	**3.0**	**-0.9**

In the EnergyPlus UCSD displacement model tested there is a need to define a heat gain distribution factor that defines the fraction of the convective heat gains that mixes into the occupied zone (as opposed to being directly transported by the plumes into the mixed upper zone, and thereby not felt by the occupants). There is a large uncertainty on the magnitude of this parameter, initial simulation were performed using the default value (0.4). Additional simulations with the heat distribution fraction value adjusted to 0.5 and showed that this change on this parameter produces significantly different temperature results (see table 6). Clearly, further research is needed to determine the adequate value of this parameter in different cases.

CONCLUSION

Adequate HVAC system sizing is a complex process involving many variables that can condition final results. The sizing results obtained for the present case show that the definition of the sizing design day and correct modeling of airflow pattern have a significant impact on system size (between 4% and 64%, depending on the case).

The CFD simulations of indoor flow fields and internal temperatures proved invaluable fine tuning of the design. The stage inflow systems tested in the CFD simulation scenarios Classical LB+LN and Modern LN+HB showed good results and confirmed that the proposed system can meet air temperature and indoor velocity design goals for the main seating and balcony area. The main concern that was initially raised by the proposed design was not confirmed by the simulation results: the stage systems do not increase the airflow velocity in the first rows of the seating area.

The comparison of predicted occupied zone temperatures results showed a good agreement between EnergyPlus and CFD. However, there is a large uncertainty in the sensible heat gains distribution fraction that must be address in future research.

ACKNOWLEDGEMENT

This work was funded by the Fundação Calouste Gulbenkian (Ph.D. Grant 126724).

The authors would like to thank Osório Tomás, Sam Wise, Michael Bradbury and David Richards.

REFERENCES

LBNL, EnergyPlus. (2010). Energy Plus Documentation: Getting Started with EnergyPlus, EnergyPlus Engineering Reference, Input and Output Reference.

da Graça G.C., Linden P.F., 2004. A simple model for heat transfer in displacement-ventilation. RoomVent 2004, 9th international conference in University of Coimbra - Portugal, 5-8th September 2004, pp 6, 7 Fig., 10 Ref.

Carrilho da Graça G. Simplified models for heat transfer in rooms. Ph.D. Dissertation. San Diego: Department of Engineering Physics, University of California; 2003.

Gates, A., Liley, B., Donn, M.. 2011. New Zealand's new weather data – How different? Proceedings of 12th International Conference on Building Simulation (Building Simulation 2011), Sydney, Australia.

http://apps1.eere.energy.gov/buildings/energyplus/weatherdata/6_europe_wmo_region_6/PRT_Lisboa.085360_INETI.zip (accessed in 02/2012).

2009 ASHRAE Handbook – Fundamentals. American Society of Heating, Refrigerating and Air-Conditioning Engineers, Atlanta, 2009.

CHAM, PHOENICS VR. http://www.cham.co.uk.

(2012).

Andreopoulos, J., Rodi, W., 1984, Experimental investigation of jets in a crossflow, Journal of Fluid Mechanics, 138, pp 93-127.

Chan, C., and Lam, K., 1998, Centerline velocity decay of a circular jet in a counterflowing stream, PHYSICS OF FLUIDS, volume 10, number 3.

H Hangan, F McKenty, L Gravel, R Camarero, Case study: Numerical simulations for comfort assessment and optimization of the ventilation design for complex atriums, Journal of Wind Engineering and Industrial Aerodynamics, Volume 89, Issues 11–12, September 2001, Pages 1031-1045.

M. Kavgic, D. Mumovic, Z. Stevanovic, A. Young, Analysis of thermal comfort and indoor air quality in a mechanically ventilated theatre, Energy and Buildings, Volume 40, Issue 7, 2008.

A. Scanlon, A. Calderone, CFD BENCHMARKING: HAMER HALL AUDITORIUM CASE STUDY, Proceedings of Building Simulation 2011, 12th Conference of International Building Performance Simulation Association, Sydney, 14-16 November.