CFD BENCHMARKING: HAMER HALL AUDITORIUM CASE STUDY

Anthony Scanlon and Anthony Calderone
Aurecon, Australia
anthony.scanlon@aurecongroup.com anthony.calderone@aurecongroup.com

ABSTRACT

This paper presents a case study of the role that Computational Fluid Dynamics (CFD) played in the design of modifications to the existing Hamer Hall auditorium Heating Ventilation and Air-Conditioning (HVAC) systems.

The use of CFD allowed the designer to benchmark the potential performance of proposed modifications against the existing installation. Further, it allowed them to anticipate areas where improvements may not be satisfactory, allowing them to explore available options and communicate this information to the end user. Therefore, building simulation allowed the designer and end user to be better informed, and thus increased the likelihood of satisfaction with outcome of the modification works.

INTRODUCTION

The Southbank Cultural Precinct in Melbourne is currently undergoing a major redevelopment of Hamer Hall, the main Concert Hall in Melbourne and home of the Melbourne Symphony Orchestra. The building was completed in 1982 and it is generally recognised that over the past 29 years it has fallen behind world standards for acoustic performance and occupant amenity. Consequently one of the main criteria in the redevelopment is to improve the thermal comfort within the auditorium for both the performers and the audience.

Anecdotal evidence from the Victorian Arts Centre staff indicated that there were thermal comfort issues within the auditorium generally revolving around cold air dumping and also areas that were considered stuffy.

A number of concepts for the upgrade to the air condition system were proposed during the design stage for the redevelopment all of varying cost and complexity. Given the constraints of the existing building and systems installed the proposed system was expected to improve the comfort issues that were raised however to assist in providing confidence in the design a Computational Fluid Dynamics (CFD) investigation was undertaken.

In addition to general rebalancing, the major proposed change is the reconfiguration of the underfloor exhaust to the front stalls area to serve as supply air and the alteration of some high level supply grilles to act as exhaust.

SYSTEM OUTLINE

The auditorium is generally divided into the following major spaces; stalls, circle, balcony, stage, wings, choir and boxes.

Existing Ventilation System

The existing system that serves the auditorium comprises two existing air handling units and two existing spill air/smoke exhaust fans.

The system layouts make use of the symmetry about the building axis line, with each being essentially the mirror of the other.

The existing supply scheme utilises 14°C supply air delivered by overhead diffusers, with supplemented by sidewall supply diffusers. The stage is cooled via supply to the wings, and by high-level grilles (above the boxes).

Return air to the air-handling units is primarily via sidewall grilles on the stall, circle and balcony levels, supplemented by ceiling return above the stage, through the side walls and underfloor return grilles in the lower stalls.

Spill air is exhausted by the smoke exhaust fans with a majority of the air drawn from ceiling level above the stage supplemented by a mid-level wall grille.

This arrangement can be seen below in *Figure 1*.

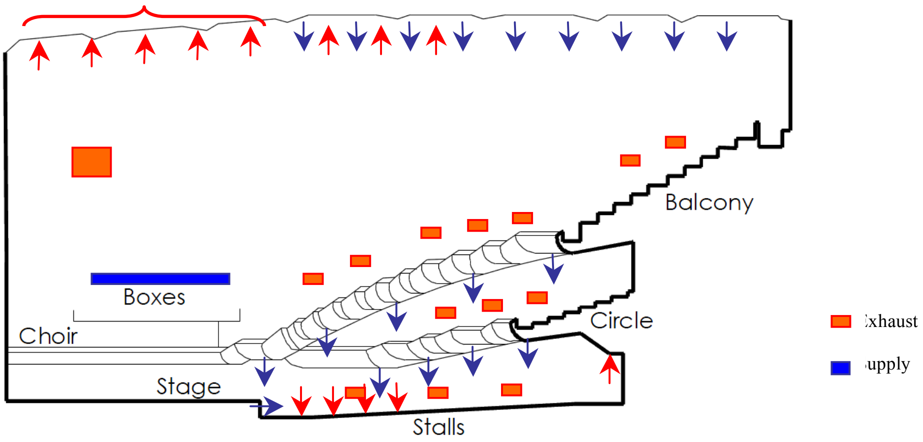

Figure 1 Existing system ventilation scheme

Proposed Ventilation System

The proposed alterations to the air distribution system result in the addition of two air handling units and two spill air fans. The proposed supply scheme for the balcony, circle and back stalls remains essentially unchanged. The supply to the front stalls is reconfigured to an underfloor displacement system, with the supply air temperature of this portion of the system raised to 18°C. The stage cooling is via high-level jet diffusers grilles (above the boxes), replacing the wall grilles, with the supply component via the wings reduced.

The sidewall return via grilles in the circle and balcony levels remains as per existing, with a rebalance in the stall areas. The underfloor return is replaced by the displacement supply system mentioned earlier, with the ceiling supply diffusers above the front stalls altered to act as return. Additional ceiling return is added above the upper balcony to supplement exhaust from the auditorium.

This can be seen in *Figure 2* below.

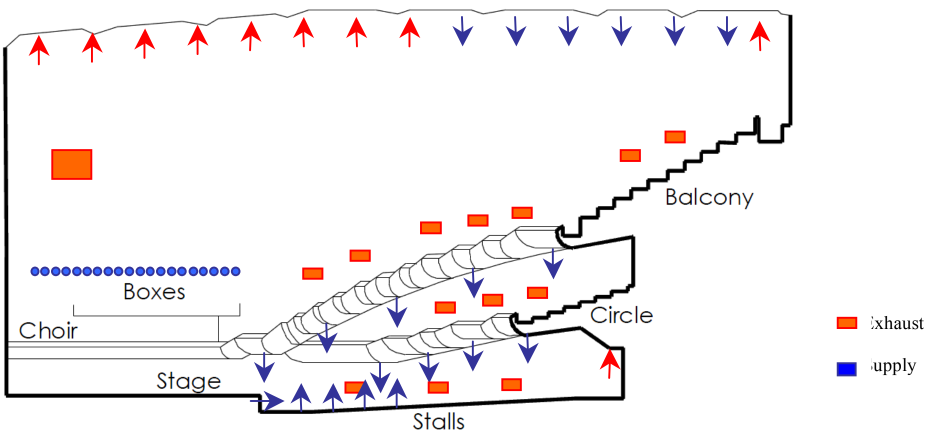

Figure 2 Proposed system ventilation scheme

SIMULATION PARAMETERS

Software

The CFD software package Phoenics was used in this assessment. Phoenics provides a full capability for grid generation, solution and post-processing via 3-D or text input. Phoenics has been validated under a number of applications, including in building environmental simulation (Gaspar, PD et al. 2003).

The Flair module of Phoenics is dedicated to CFD airflow visualisation and analysis applications within the built environment and provides the following capabilitie; steady-state and transient CFD analysis, turbulence models (including the k-e model), buoyancy models, conjugate heat-transfers, a library of materials based on CIBSE reference data and a range of HVAC-related objects, including diffusers and fans.

Modelling Assumptions

Computational fluid dynamics is a time intensive process. In order to reduce the model development and calculation time for this complex simulation, it is essential to make simplifications to allow convergence of the simulation within a finite time. Further, as Phoenics makes use of a structured grid angled and curved geometry can result in increased convergence times if not properly considered.

The following table contains a list of simplifications, and comments on the justifications:

- Angles and curves in the model were represented by a series of step elements fitting with the grid, this included coordinate system transformation to correct for curvature of rows. Spaces that are not connected to the space, but are technically present within the domain (such as projection room, ceiling voids, etc) are treated as voids and not included in the calculation
- Minor realignment of geometry to create common faces
- Asymmetries about the building axis line were not considered significant, and thus ignored, for instance off-centre seating
- The implemented size of slots/grilles/diffusers varies from nominal to suit grid
- The effect of humidity was neglected
- The initial parameters represent condition in the performance space after the audience has settled, and thus subject to negligible transients in heat loads

Grid Generation

The grid spacing implemented in the model was 167x140x92, or 2,150,960 cells. This resulted in an effective resolution of approximately 0.25-0.3m. Note that the grid was able to expand where little detail was required (generally only in the z direction above the balcony zone), such that resolution in matched the nominal unit size in occupied areas.

Generally, this is a coarse grid given the distribution of supply, exhaust and loads through the space. However, increases in resolution were not feasible due to the limitations of 32-bit computing; sensitivity analysis was undertaken of test spaces representing half model. This analysis was undertaken to determine the impact of the course grid. It involve creating a small model of a diffuser with two separate grids, one a course grid and the other a fine grid. The difference between the two models was compared in this sensitivity analysis and no significant difference was observed.

Occupants

Occupants were modelled as blocks heat sources composed of domain material (in this case air) with a fixed heat flux, occupying a seat that provided obstruction to air flow. Individual members, with seat, occupied a space of 0.5m by 0.5m by 1.2m high. They were considered effectively "at rest", and thus assigned a 65W sensible load.

On the stage equipment and performers were represented by a fixed 60W/m² heat load with 20 W/m² heat load in the wings.

Obstructions

Obstructions, such as walls, ceilings and rows were considered to have sufficient thermal mass to be insensitive to local conditions in minor time scales, thus they were given a fixed surface temperature due to thermal history (pre-performance conditioning).

Ventilation System

Cooling mode operation under full occupancy was chosen for this analysis. Perfect commissioning is assumed when assigning flow rates to HVAC elements, in that nominal flow values are achieved. The system is represented without control systems, and thus unable to correct flows or temperatures in a way that may be relied upon in operation.

Note that in comparison to heating mode and isothermal cases, it has been found that turbulence models are generally poor predictors of ventilation flow in cooling mode (Einberg, G et al 2005), with the standard k-ε turbulence model provides adequate performance (Cao, G 2007) and was thus adopted for this simulation.

Diffusers

Given the coarseness of the grid, a simple diffuser type was used based on a grille/nozzle model. The actual diffuser type was simulated by selecting face velocity, aspect and angle to appropriate values depending on whether the diffuser was a ceiling-mounted register, linear slot, underfloor or wall register.

It is not expected that proper diffusion will be achieved throughout the model, but both models will be subject to the same limitation and thus are comparable.

Return air

The return air outlets were represented in the model by square face fans, which function by extracting air from the model at points. The volume of air for a particular fan was determined by the chosen face velocity.

RESULTS

As discussed above two modelling scenarios were investigated. One being the existing system as currently designed and operating, the other being the system as proposed by the mechanical engineers. The purpose of the modelling was to determine if the proposed solution was going to solve some of the system problems and also to identify if there are additional areas that require further design review and consideration.

Results of the two models are shown below for temperature and velocity in plan view at the stalls level (the area with most complaints) and in section view through the centre of the auditorium.

Temperature Plot Elevations

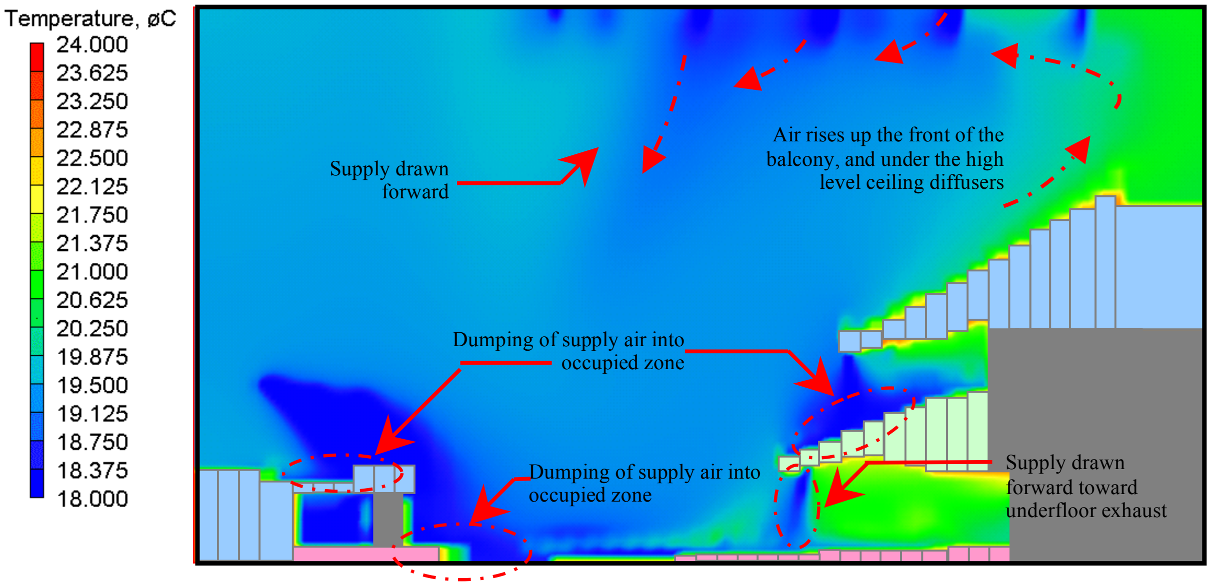

Figure 3. Elevation showing Existing Conditions

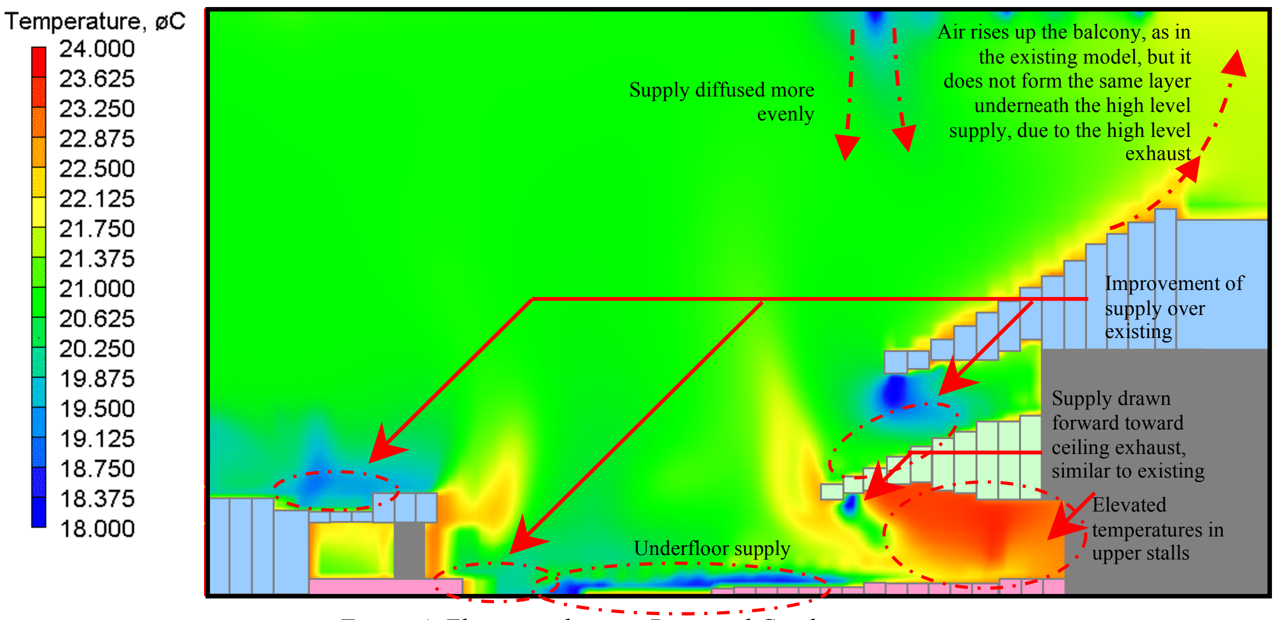

Figure 4. Elevation showing Proposed Conditions.

From *Figure 3* and *Figure 4* above we can see that with the existing conditions model the temperature conditions are generally colder and there are sharper temperature gradients when compared to the proposed conditions model. Elevated temperatures are observed in the back stalls.

In the back stalls air is not mixing as well in the proposed scenario and stagnation is occurring. This result has therefore highlighted a general improvement in comfort conditions across the stalls however an area where additional design input was required in the back stalls.

Temperature Plot Plan View

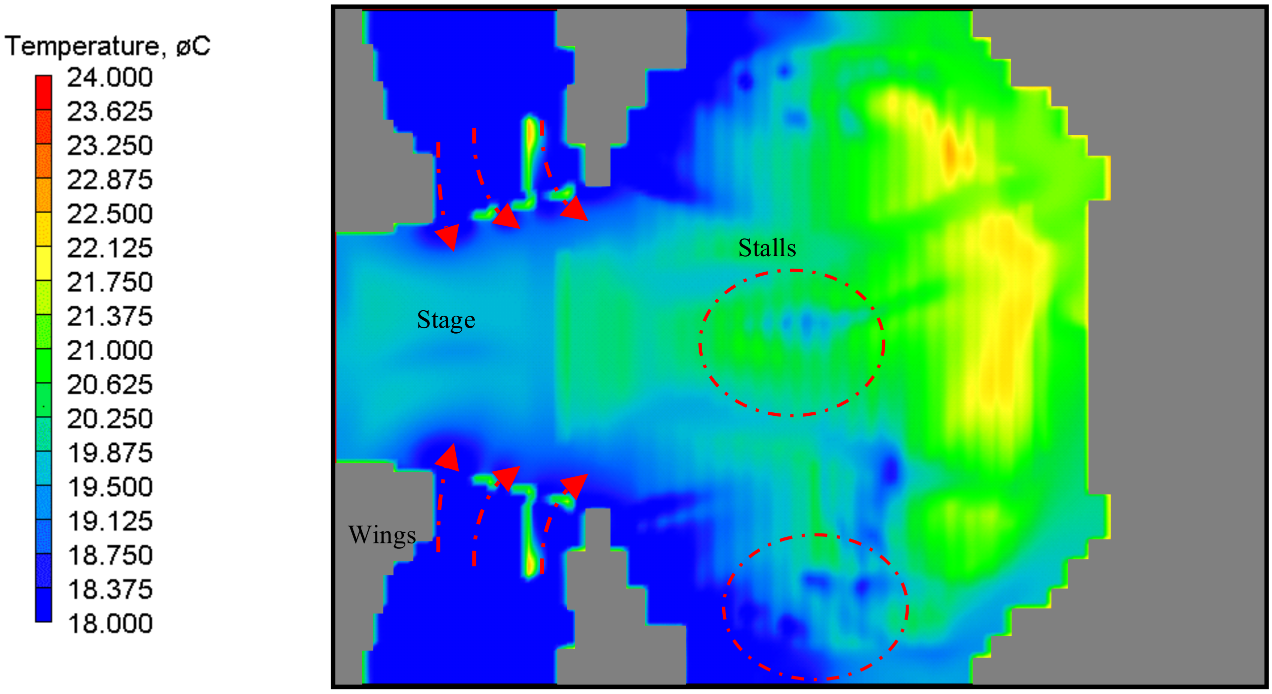

Figure 5. Plan View showing Existing Conditions

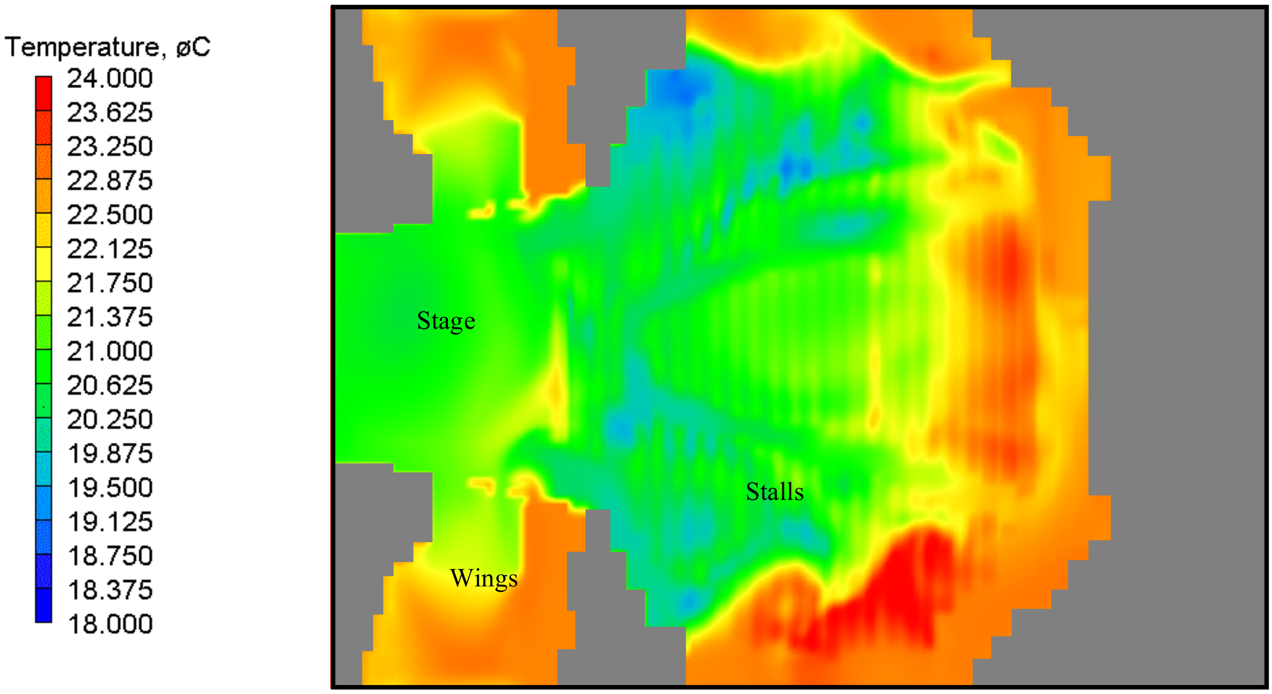

Figure 6. Plan View showing *Proposed Conditions.*

From *Figure 5* and *Figure 6* above as with *Figure 3* and *Figure 4* from the previous section we can see that with the existing conditions model the temperature conditions are generally colder. Further to this sharper temperature gradients as highlighted in the circles above can be seen in the existing conditions model when compared to the proposed conditions model. The temperature gradients in the existing conditions model are generally in excess of 1.5oC, whereas in the proposed model they are 1oC or less.

As noted above elevated temperatures are observed in the back stalls however higher temperatures were also observed in the wings. In the wings the supply air volume was substantially reduced in the proposed model which accounts for the higher temperatures.

Velocity Plot Elevation View

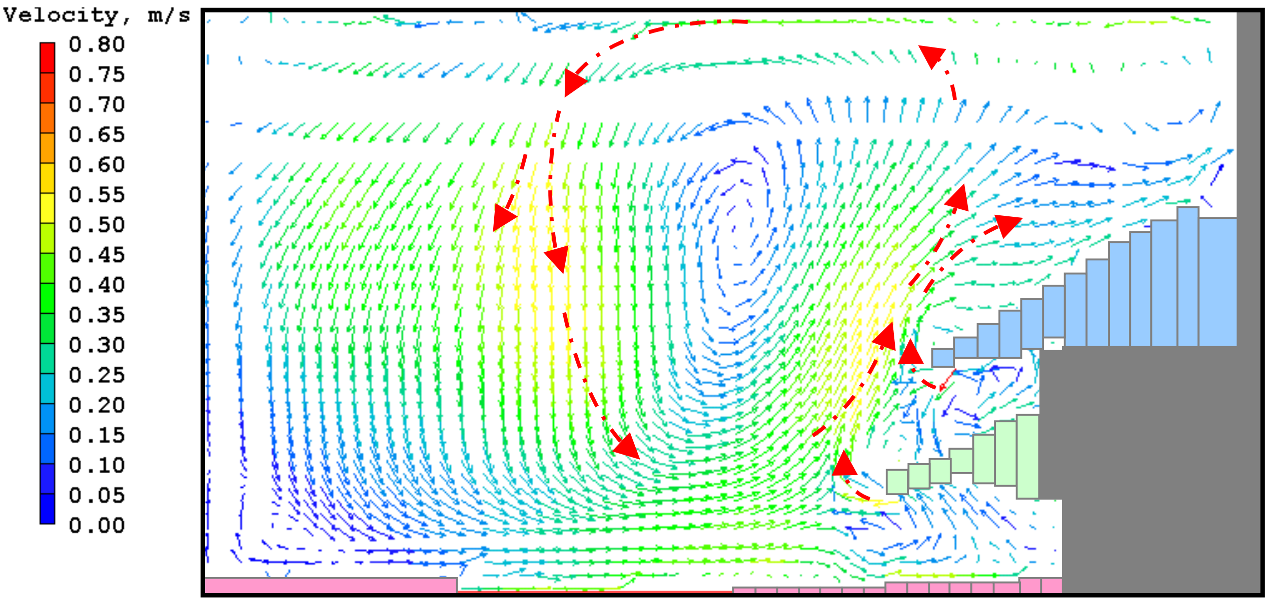

Figure 7. Elevation View showing *Existing Conditions*

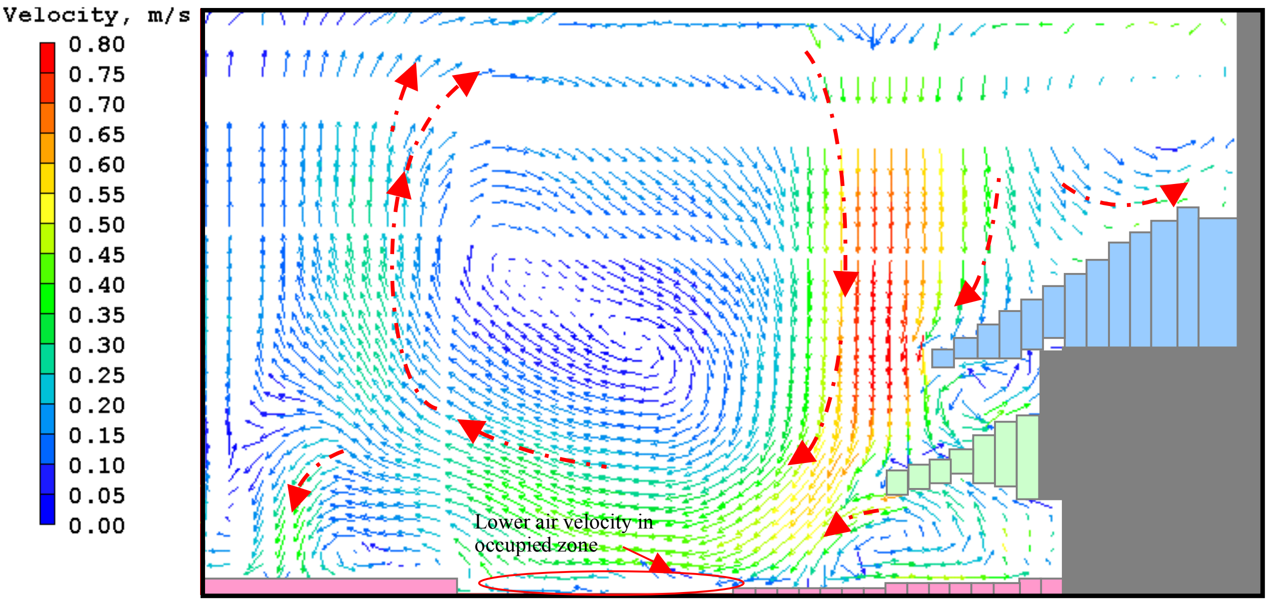

Figure 8. Elevation View showing *Proposed Conditions.*

Figure 7 shows the existing conditions which appear to correlate with the anecdotal evidence from the facility staff. This is where cold air supply is felt in the front stalls section. Air velocities of over 0.3m/s are observed in this area which are considered to be above what can typically be felt (ASHRAE Fundamentals).

Also in Figure 7 we ca see that the cold air supply from above the balcony does not appear to be directed down to the balcony area, rather it is entrained into the anti-clockwise air diffusion pattern within the auditorium area above the stalls.

The proposed system as outlined in Figure 8 shows a very different pattern. This is where the down draft at the front stalls which was understood to be the cause of most complaints is removed. Similarly air velocity in the occupied space is now below a noticeable level of 0.3m/s.

In addition to this the air diffusion over the balcony appears to now supply air directly to the balcony rather than being entrained into the front stalls area.

Velocity Plot Plan View

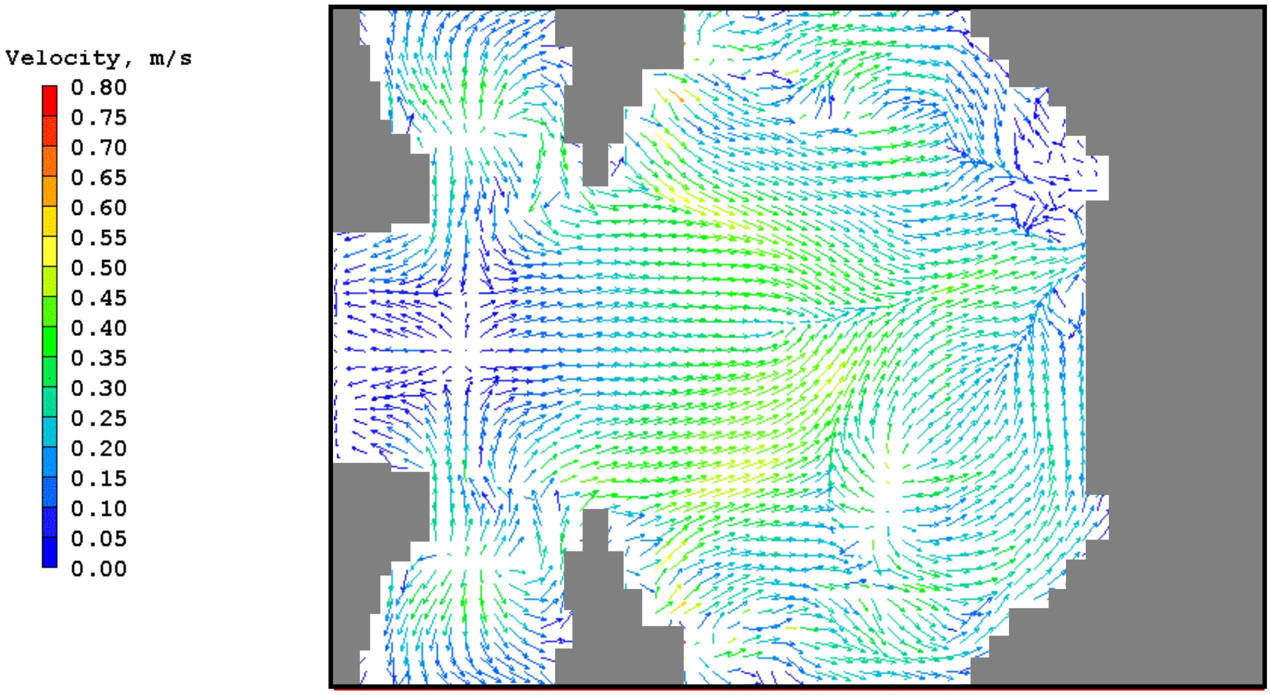

Figure 9. Plan View showing Existing Conditions

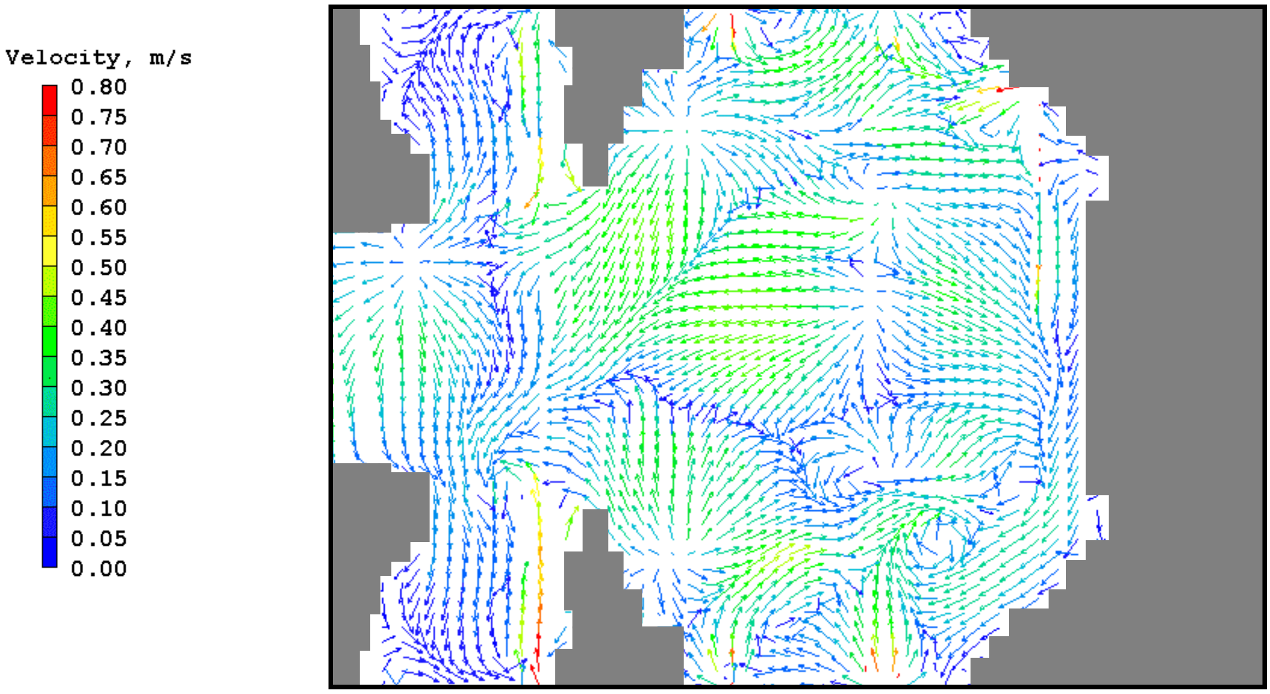

Figure 10. Plan View showing Proposed Conditions.

From *Figure 9* above it can be seen that the air flow is generally from the stage to the back stalls. This is seen better in Figure 7 above where an anti-clockwise flow pattern is observed. When comparing this to the flow pattern to that in *Figure 10* it can be seen that the existing condition flow pattern is broken up and there is no real discernable pattern. This highlights that one of the design objectives of preventing air dumping is achieved.

In addition to this it can be seen that the air velocities in the proposed conditions scenario are generally lower which is also expected to contribute to improving the thermal comfort within the auditorium.

DISCUSSION

Temperature

It should be noted that the relative rather than absolute temperature values are important in interpreting the simulation results.

Both the existing and proposed simulations show elevated temperature in the back stalls. This is partly due to the inaccuracies of the CFD model where a finer grid is not possible within the capability of a 32bit computer and partly due to the current grille selections and locations. The current grille selection results in the poor mixing of supply air in the space and easy entrainment of the supply air into the air rising through the auditorium void space for exhaust.

The grille selection used in the current CFD model is based on acoustic requirements where sound levels are desired to be low (meaning grilles that do not mix air significantly and create regenerated noise), resulting in the air not being properly mixed with the air in the occupied zone.

The CFD model has therefore highlighted an area of further design consideration. One design solution for this would be to provide a grille with a higher induction ratio. Another is to provide a new exhaust point in this back stalls area.

Whilst these two proposed initiatives are expected to improve the thermal performance in the back stalls and circle area preliminary advice from the acoustic consultant has indicated that a high induction type grille cannot achieve sound levels recommended for a concert hall. Given this requirement the additional exhaust louvers were provided to create a slight improvement to thermal comfort.

Airflow

Both systems achieve similar local airspeeds in the occupied zone, however the proposed system appears to have air speeds which are closer to the acceptable threshold of 0.3m/s.

The simulation of the existing system indicates the formation of a circulation pattern which may result in large scale dumping of air above the front stalls and short-circuiting of the stalls high level supply air to the high level extract air. The airflow sweeping up from the stalls (as a result of the heat from the audience) also appears to divert supply air from below the balcony and circle and prevents the overhead supply to these areas. This observation in the CFD results is similar to the anecdotal evidence of actual air patterns in the auditorium experienced by users.

The proposed case also shows the presence of a circulation pattern; however in this case the pattern is confined to the void above the stalls seating and high-level air dumping appears less likely to occur.

LESSONS LEARNT

- The use of CFD modelling for comparison of an existing system to a proposed system has proved useful in validating a mechanical design concept and has also highlighted areas where further design review was required.
- The process is time consuming however given the importance of the system in achieving thermal comfort within budgetary and acoustic design constraints has been considered useful.

CONCLUSION

The results of the CFD analysis indicated that the proposed alterations generally achieve the following when compared to a model of the existing:

- Delivery of supply air from diffusers to the front stalls and balcony appears to improve
- Circulation patterns in the void space appear to be more moderate and hence air dumping may be less likely to occur
- Local temperature variations may be more moderate

It should be noted that in the CFD simulation, both the existing and proposed systems demonstrate stagnation of air and higher temperatures in the back stalls and circle area. Feedback from users of the auditorium would also support this finding of temperature in these areas. The solution adopted to overcome this problem was to increase exhaust flows from this area in addition to the replacement of the existing linear slots with an additional row of slots against the back wall of the stalls and circle levels.

REFERENCES

Cao, Guangyu (2007) Turbulence models validation in a ventilated room by a wall jet. Ene-39.40533 Postgraduate seminar on energy technology. Laboratory of heating, ventilation and air-conditioning, faculty of mechanical engineering, Helsinki University of Technology.

Einberg, G., K. Hagstrom, P. Mustakallio, H. Koskela, H. and S. Holmberg (2005) CFD modelling of an industrial air diffuser – predicting velocity and temperature in the near zone. Building and Environment. Volume 40, Issue 5.

Gaspar, PD et al. (2003) Performance Evaluation of CFD Codes in Building Energy and Environmental Analysis. Eighth International IBPSA Conference, Eindhoven Netherlands.