# INNOVATIVE DESIGN SOLUTIONS FOR BURN INTENSIVE CARE UNITS

Isabelle Lavedrine[1], Patric Thomas[1], James Tharp[2], Jerry Sipes[2]
[1]Arup, 901 Market Street, Suite 260, San Francisco, CA 94103
[2]Price Industries, 2975 Shawnee Ridge Court, Suwanee, GA 30024

## ABSTRACT

Burn Intensive Care Units (ICU) have among the most stringent design criteria for patient rooms in hospital design. Communication between the healthcare professionals, the architect designing the room layout and the mechanical engineer designing the HVAC system is critical to ensure that their design converges to meet the required therapeutic criteria. A Computational Fluid Dynamic (CFD) analysis played an important part in this process for a University Medical Center, and a physical test of the final set-up helped to fine-tune and confirm the design. Further comparisons between the CFD analysis and the experimental measurements were carried out to validate the original numerical approach.

## INTRODUCTION

In a Burn ICU, the patient is weak and sensitive due the high body surface temperature caused by the burns, and is susceptible to infections and fluid loss. To ensure comfort for the patient and promote healing, strict design criteria are to be met for both air speeds and temperatures.

The design criteria for the studied Burn ICU were determined based on the California Building Code, in particular the California Mechanical Code 2001, on guidelines from the AIA (AIA 2001) and ASHRAE (ASHRAE 2003), as well as discussion with the clinical staff. The criteria specified that velocity at the patient was not to exceed 0.25m/s (50fpm – feet per minute), and the environmental temperature around the patient was to be kept between 21.1°C and 29.4°C plus or minus 1°C (between 70°F and 85°F plus or minus 2°F). Positive pressure needs to be maintained inside the room to minimze infiltration of possible infectious particles. Relative humidity needs to be kept in the range of 30 to 60%, with higher humidity preferred to minimize the fluid loss of the patient. It is the role of the mechanical engineer to ensure that these criteria are achieved inside the Burn ICU. Therefore communication with the architect who designs the room layout is crucial.

The proposed ventilation system design for Burn ICU relied on providing supply air at high level, and extracting mainly at low level near the entry door, utilizing a laminar flow type ventilation system which aims to provide vertically downward flow conditions.

CFD analyses have been used extensively by Arup and others to examine the performance of ventilation systems (Lavedrine & Woolf, 2003 – Woolf, 1999), and specifically for very controled environments such as Operating Surgical Rooms (Memarzadeh & Manning, 2002). There are also many design guides for tighlty controled patient room ventilation systems, often from the manufacturers of the system components (Titus, 2003 – Price, 2004).

## NUMERICAL SIMULATIONS

In the considered University Medical Center, a typical single-patient room is approximately 4.4m by 5.3 m by 2.6m (14.5ft · 17.5ft · 8.5ft). The room has two windows, each 1.2m by 1.2m (4ft · 4ft). The patient bed is approximately centered inside the room. Two exhaust grilles are present in the room, at low level close to the entry door. Supply grilles are at high level in the vicinity of the bed. Additionally, at high level close to the bed can be found the lighting fixtures as well as several IV tracks.

**Original Unit Set-up**

- Description

The first layout proposed by the architects was such that above the patient bed, supply grilles alternated with overhead radiant heating panels, while large ceiling lighting panels were set around the bed. While this layout provided that good lighting close to the bed, the strict velocity requirements inherent to a Burn ICU could not be achieved without excessive air volumes and associated air turbulence at the bed level.

- CFD model and results

A CFD model was built assuming summer conditions outside. The internal wall surface temperatures were taken from the results of the dynamic thermal analysis of the space using Arup’s in-house simulation program ROOM. The thermal analysis modeled the room as a well-mixed zone. All solar gains and radiation gains are modeled in the ROOM model and represented in the surface temperatures obtained from it. Therefore the CFD simulation does not include any radiation modeling directly, simplifying the model set-up and reducing the length of the calculation process. The predicted surface temperatures are shown in Table 1:

*Table 1*
*Surface temperatures*

| SURFACE | TEMPERATURE IN C ( F) |
|---|---|
| All walls (including ceiling and floor) | 22.2 (72.0) |
| Windows | 31.4 (88.5) |
| Radiant Panel | 32.2 (90.0) |
| Bed | adiabatic |

The simulation was carried out using the CFD code StarCD v3.20. The computational domain consisted of approximately 125,000 cells. The total volume of the modeled space was just over 56 $m^3$ (2,000$ft^3$). The average cell size is approximately 0.1m (4in). The mesh was further refined along the walls, as well as close to the supply and exhaust grilles.

The other assumptions for the simulation were as follows:

- The flow was assumed to be steady-state, i.e. it represented a fixed moment in time at which conditions are estimated to change very slowly and/or remain constant for a long period of time.
- The lighting load was assumed to be 21.5W/$m^2$ (2W/$ft^2$), 50% of which was convective, giving a total convective load of 53.3W.
- The occupant load was assumed to be 25W convective. The patient was represented as a fluid heat source lying on top of the bed.
- The total air supply was 0.247$m^3$/s (525cfm – cubic feet per minute), and the supply air temperature was set to 19.4 C (67 F).
- Air was extracted evenly at low level.
- The radiant panel was just under 1.4m².
- A standard high Reynolds number k-e turbulence model was used.
- Gravity was taken as –9.81$m^2$/s.
- There was no other load in the space apart from people and lighting.

The supply grilles were modeled by representing the real exit velocity over the overall area of the grille. For this model, manufacturer's data were used that the perforated grille was 13% open (i.e. its free area was 13% of its total area), giving an exit supply velocity of 1.294m/s (255fpm). It is important to model the correct velocity in order to represent the vertical throw of the diffuser properly.

Figure 2 below shows the geometry set-up of the CFD model.

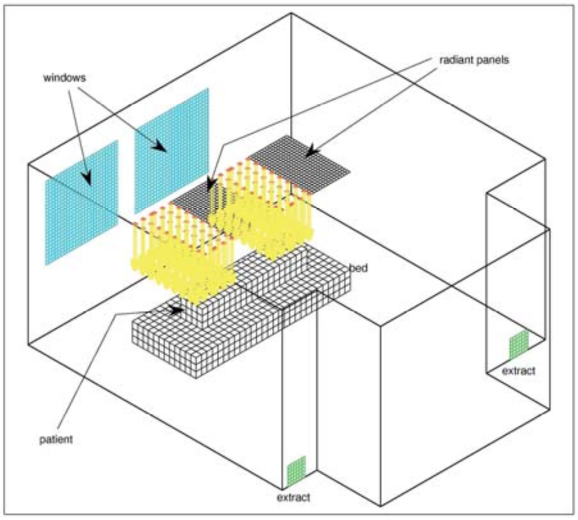
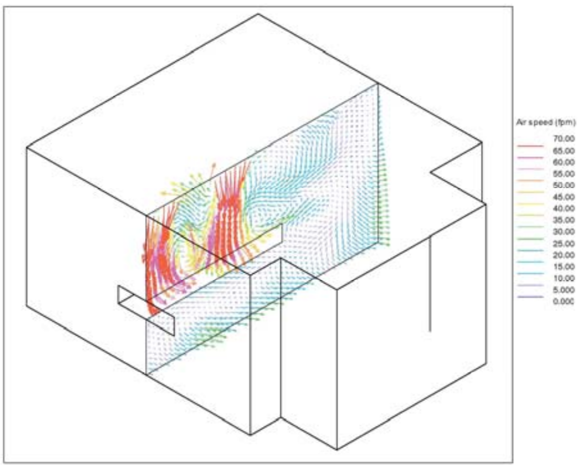



*Figure 2 CFD geometry*

No mesh sensitivity analysis was carried out. The aim of the CFD model was to give qualitative results on the appropriateness of the designed room layout, as well as an indication of the air speeds close to the patient. Figure 3 shows the velocity vectors and the temperature contours in a section in the middle of the bed. It is quite clear from the plots that there are significant variations in temperature close to the patient, as well as air speeds higher than the design criteria. The predicted temperatures close to the patient vary between 20°C at the head of the bed to 21.5°C at the foot of the bed (from 68°F to 71°F), and the predicted air speeds vary from 0.08m/s to 0.35m/s (15fpm to 69fpm).



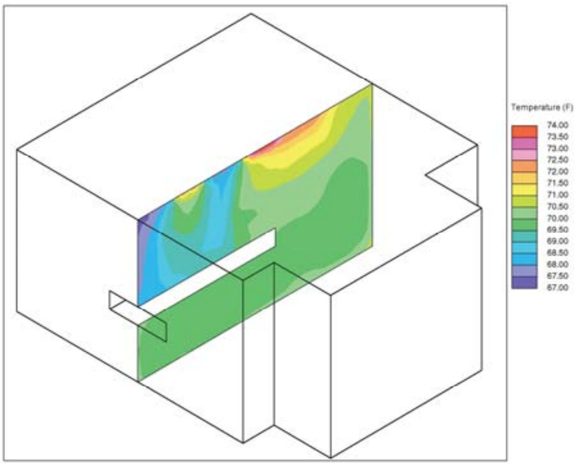



*Figure 3 Velocity vectors and temperature contours*

**Modified Unit Set-up**

Following the presentation of the CFD results to the architects for the first set-up, a modified layout was adopted. The architectural drawing can be seen in Figure 4.

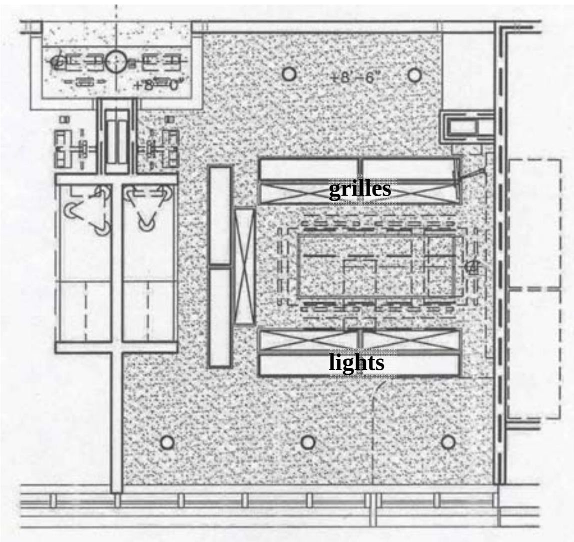



*Figure 4 Architectural drawing of Burn ICU layout*

The supply grilles were positioned around the bed in order to create an “air curtain” which keeps the air inside the patient zone constantly changing to be as clean as possible, while at the same time providing air speeds lower than 0.25m/s.

- Description

The model was of the same size as the previous model and most of the assumptions are similar. However, the quantity of air supplied was reduced in order to reduce the supply velocity, and the area of the radiant panel was increased to cover an area equivalent to the patient bed at ceiling level:

- The total air supply was 0.215m$^3$/s (455cfm), 30% at the foot of the bed, 35% on each side.
- The radiant panel was just over 1.85m$^2$ (20ft$^2$).
- An additional extract was added a high level close to the window and the bathroom corner.

Figure 5 shows the geometry of the CFD model.

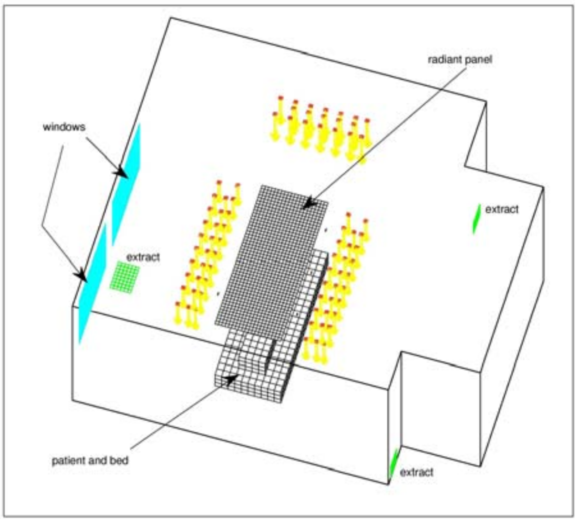
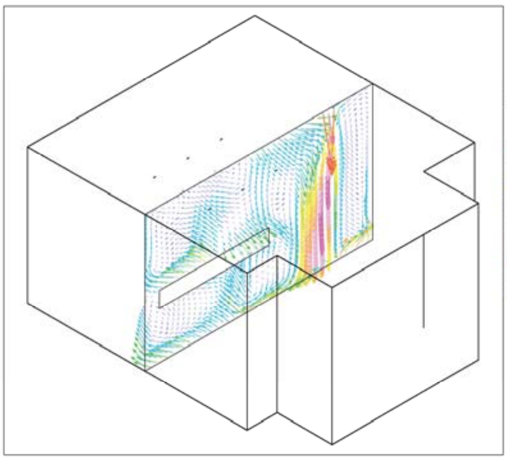



*Figure 5 CFD geometry*

- CFD model and results

As for the previous model, the supply grilles were modeled by representing the real exit velocity over the overall area of the grille. The open area of the grille was assumed to represent 13% of the total area, giving an exit supply velocity of 1.04m/s (205fpm). The distance between the edge of the bed and the edge of the supply grille was 0.28m (11in) in the CFD model. Figure 6 shows predicted velocity vectors and temperature contours in a section through the middle of the bed. The air speed above the patient did not exceed 0.25m/s (50fpm), and the predicted temperature was very uniform around 21.1°C (70°F).



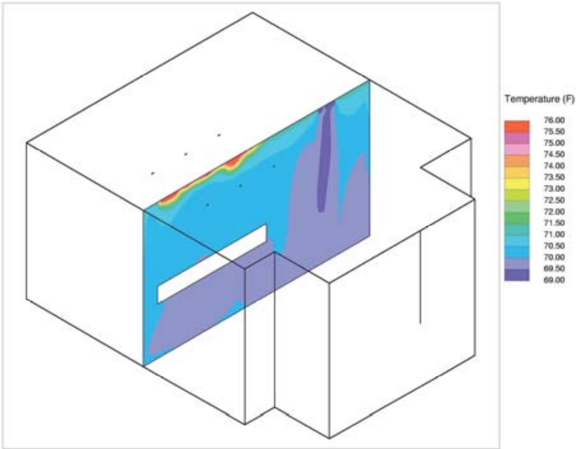



*Figure 6 Velocity vectors and temperature contours*

From the CFD results, it was recommended that the distance between the edge of the bed and the edge of the supply grille be not increased above 0.3m. This could create a zone of more stagnant air around the patient's bed which was not desirable.

With this configuration of diffusers and given flow rate, it was not recommended either to decrease the distance between the edge of the bed and the edge of the supply because velocities higher than 0.25m/s (50fpm) would then occur in the vicinity of the patient. However, should the free area of the supply grille change (to a higher percentage than 13%), for example, due to the introduction of HEPA filters, then it might be possible to reduce this distance and obtain still air speeds within the desired range.

Finally, the diffuser at the foot of the bed did not provide any benefit for the patient, nor enhanced the "air curtain" around the bed which could be achieved without this additional grille. It was therefore recommended to remove this grille in the final room layout.

An additional CFD run was carried out with increasing the supply air temperature to 32.2°C (90°F), due to concerns about the ability of maintaining the air curtain around the patient bed with the increased buoyancy of the flow. For this case, the predicted velocities above the bed are vey low, and were found to be at the limit of feasibility. Therefore a maximum supply temperature of 29.4°C (85°F) might be more practical.

## EXPERIMENTAL MOCK-UP

### Description of mock-up

The purpose of the mock-up was to verify the airflow patterns around the Burn unit bed. The mock-up was set-up in the Suwanee, GA, office of Price Industries. Figure 7 shows the set-up and the diffuser arrangement. Air motion was measured using a sensor tree. The diffusers used were Price LFDCs with HEPA filters. Smoke tests were also performed and recorded on videotape.

*Figure 7 Diffuser arrangement in the ceiling and general set-up*

All measurements were taken with isothermal air to reduce temperature effects as much as possible. The sensor tree was used to take temperature and air velocity measurements at 15cm (6in) intervals throughout the entire LFDC pattern. The sensor tree was moved to each position and the data averaged over one minute to produce the results presented below.

### Tests

The test were performed for a variation of conditions, where the supply flow rate was kept constant at 0.28m³/s and the air supply temperature was varied. Also, the test were carried out first without the radiant panels, which were switched on for a second series of measurements. Table 2 summarizes the tests carried out:

*Table 2*
*Tests*

| TEST | AIR SUPPLY TEMPERATURE IN C ( F) | RADIANT PANEL |
|---|---|---|
| 1 | 21.1 (70) | -- |
| 2 | 29.4 (85) | -- |
| 3 | 32.2 (90) | -- |
| 4 | 21.1 (70) | ✓ |
| 5 | 29.4 (85) | ✓ |
| 6 | 32.2 (90) | ✓ |

To assess the impact of the radiant panel, radiant temperature sensors were placed on the center of the bed. These consisted of a copper sphere approximately 7.5cm (3in) in diameter and painted black. A thermocouple was inserted into the sphere to measure the temperature without the influence of the surrounding air temperature. The radiant temperature sensor and the arrangement on the bed is shown in Figure 8.

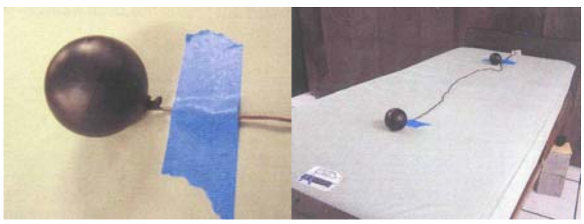


*Figure 8 Radiant temperature sensor and arrangement on the bed*

### Results

The modified unit set-up, following the recommendations from the CFD analysis, placed the diffusers at 0.6m (24in) from both sides of the centerline of the bed (i.e. 0.3m or 12in from the edge of the bed). The experimental smoke tests showed that this dimension worked for 21.1 C (70 F) supply air and provided efficient air flow over the surface of the bed. However, when 29.4 C (85 F) supply air was introduced, the diffuser placement did not provide proper air displacement over the patient bed. To do so, the diffuser placement had to be revised to approximately 0.45m (18in) from both sides of the centerline of the patient bed for efficient air flow.

Figure 9 present some of the measured data, i.e. the air speeds in feet per minute above the bed (height in inches). These measurements were taken in positions such that they were made directly above the bed and as close to the center of the bed as possible.

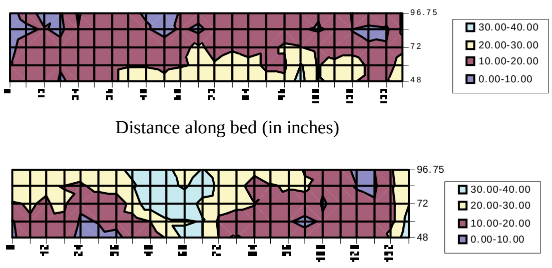



*Figure 9 Airflow patterns for 600CFM at 70 F and no radiant panel*

It was noticed that for all the measurements, the velocities in the measured sections increased when the radiant panel was switched on. Also, the air flow pattern over the bed was more stable, i.e there was less variation in air speed between the two measured sections with the heaters operating. Figure 10 presents some of the results with the radiant panel switched on.

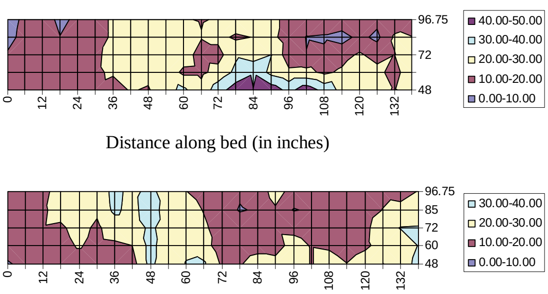



*Figure 10 Airflow patterns for 600CFM at 70 F with radiant panel*

The other advantage of the radiant panel was to provide higher temperatures close to the bed, especially when the air supply was 21.1°C (the resultant temperature close to the bed was measured on average at 27.2°C - 81°F). The radiant panel would therefore enable the room air temperature to be decreased while keeping the patient warmer providing greater comfort to the medical staff and reducing air reheat requirements.

With this configuration, the experimental data shows that the air motion is very low (less than 0.1m/s, or less than 20fpm) around the patient area.

## COMPARISON BETWEEN NUMERICAL AND EXPERIMENTAL RESULTS

### Description

A comparison between the results of the mock-up and CFD analysis was undertaken in order to validate the original numerical approach. It was not the intention to obtain close matching data between the two methods given that the CFD model was not a representation of the mock-up. Validation was expected in the form of similar values of air speeds close to the bed. Therefore only one case was considered, where the supply temperature is 21.1°C (70°F) and the radiant panel was not switched on.

The CFD model was modified from the model used to represent the Modified Unit set-up. The supply grilles were positioned so that their edge matched the bed's edge in plan. Also, the inlet representing the grilles was modified so that the supply exit velocity took into account the larger open area of the diffusers with HEPA filters (50%), and the area on which the velocity was applied was increased to match 50% of the total diffuser area. Furthermore, the following were changed:

- The patient heat source was removed from the room.
- All walls and surfaces were taken as adiabatic, in particular the radiant panel.
- The supply grille at the bottom of the bed was removed.

Finally, the mesh was refined uniformly between the ceiling and the bed, bringing the number of cells in the model to just under 300,000.

Figure 11 shows the CFD geometry.

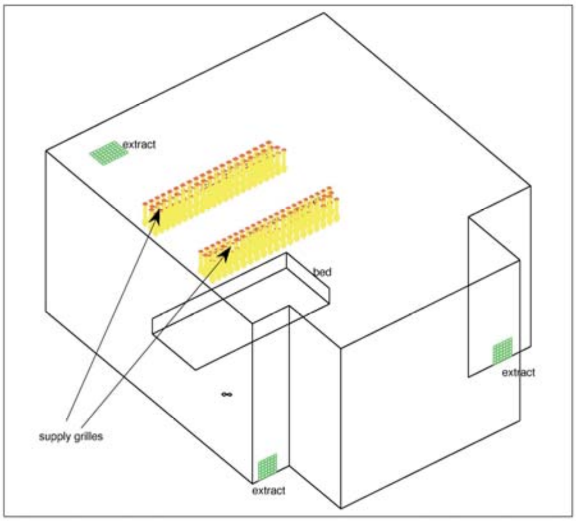



*Figure 11 CFD geometry*

**Results**

The CFD model was run in pseudo-transient mode, and it took 125 seconds to achieve convergence. Figure 12 shows predicted velocity vectors and temperature contours in a section through the middle of the bed. The air speed above the patient did not exceed 0.25m/s (50fpm), and the predicted temperature was very uniform around 21.1°C (70°F). The maximum velocity predicted over the bed is 0.187m/s (36.8fpm), which closely match the maximum velocity recorded over the bed in the mock-up (0.194m/s – 38.21fpm).

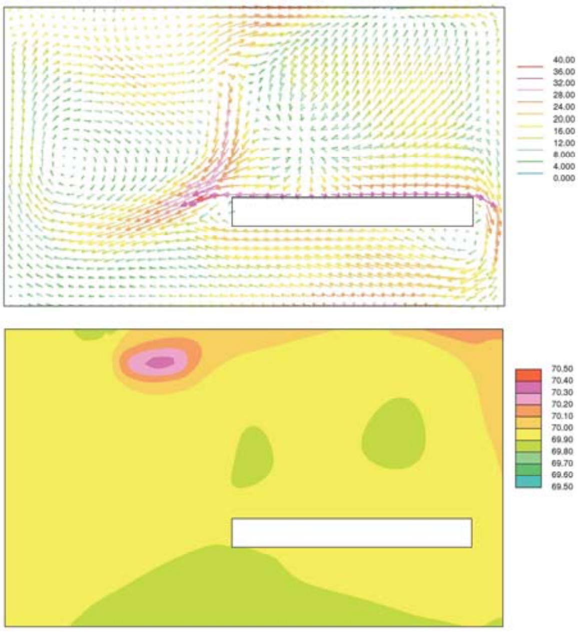


*Figure 12 Velocity vectors and temperature contours*

## CONCLUSIONS

The combined numerical analysis and experimental mock-up approach was the founding basis to obtain a better design for the patient. The initial CFD analysis enabled better communication of the mechanical engineering design intent to the architect. Following the redesign of the room layout, the revised CFD model showed that the predicted velocities close to the bed (and hence the patient) would not exceed the design criteria. The CFD results also showed that the supply diffuser grilles could not be positioned too far from the bed, and that the diffuser grille at the bottom of the bed was redundant. These conclusions enabled the number of experimental tests and measurements to be significantly reduced.

In order to achieve full validation of the CFD results, several parameters could be modified for a sensitivity analysis, such as variation in the turbulence intensity and length scale set for the inlets representing the supply grilles, or even the set-up of the inlets themselves. Alternatively, a CFD model representing the exact experimental set-up could be be created and run.

## REFERENCES


California Building Code, California Mechanical Code. 2001

The AIA Academy of Architecture for Health. 2001. “Guidelines for Design and Construction of Hospital and Health Care Facilities”.

ASHRAE Special Project 91. 2003. “HVAC Design Manual for Hospitals and Clinics”.

Lavedrine I., Woolf D. 2003. “CFD analysis in the Built Environment – Internal ventilation”. In *QNET-CFD Network Newsletter, Vol2, No2*, Thematic Network for Quality and Trust in the Industrial Applications of CFD, European Union.

Woolf, D. 1999. “Built environment simulation using CFD”, *Institute of Mechanical Engineers Seminar on Computational fluid dynamics: Technical developments and future trends*, London.

Memarzadeh F., Manning A.P. 2002. “Comparison of Operating Room Ventilation Systems in the Protection of the Surgical Site”, *2002 ASHRAE Annual Meeting*, Honolulu, HI.

Titus Product Catalog. 2003.

Price Air Distribution Products. 2004.

STAR-CD Methodology Manual, Version 3.20, CD-Adapco, 2003.