Safety Engineering Series p. 44 - 48, DOI 10.2478/v10281-012-0004-y

USE OF CFD TOOL ANSYS FLUENT FOR FIRE SAFETY IMPROVEMENT OF AN INDOOR SPORTS ARENA

Ondřej ZAVILA[1]

Research article

Abstract: The focus of the article is the design of a HVAC (heating, ventilation and air-conditioning) system for the main central space of a real multifunctional indoor sports arena. Numerical simulation of the air flow provided a basis to determine the optimal number of fire ventilators, their capacity and placement, and also to determine the maximum air-flow velocity in openings designed for evacuation in the event of an emergency. The ANSYS Fluent 13.0 software one of the world's most sophisticated CFD (computational fluid dynamics) codes was used for the numerical simulation. In this case, the specific problem of a turbulent flow in a complex geometry was under investigation. This study should serve as a base for future installation of the system in the indoor sports arena.

Key words: CFD, HVAC, numerical simulation, fire protection, indoor sports arena.

Introduction

The fire safety and security of old indoor sports arenas may present a problem that should be addressed. Many such buildings designed in accordance with previous, superseded civil engineering standards do not comply with the requirements of present fire protection standards, especially those regarding safe emergency evacuation and firefighting activities.

The indoor sports arena concerned was built in the 1980s. Its main central space provides a multifunctional use for a wide range of sport and cultural activities for up to 10 000 visitors. No HVAC system ensuring sufficient ventilation of the central area in case of fire has been installed. For this reason, the analysis was developed focusing on fire ventilation possibilities. A new HVAC system based on the analysis is to be installed in the object in the near future.

Materials and methods

The goal of the analysis was to design an optimum HVAC system for the central space of the multifunctional indoor arena and to make a numerical simulation of its function in complex geometrical conditions of the arena interior.

The HVAC system must ensure air exchange throughout the inner space of the arena whereas the air-flow velocity must not exceed 5 $m.s^{-1}$ in the inlet air openings as prescribed by the CEN/TR 12101-5 standard. All potential emergency exits and escape routes (gates, corridors) are considered air inlets. The priority of the system is to protect people in the auditorium and escape routes from the effects of heat and smoke in case of fire.

The ANSYS Fluent software was chosen for the numerical simulation of the problem. It is one of the CFD codes, i.e., programs or algorithms designed to solve primarily fluid mechanics problems (Bojko, 2008; Kozubková, 2008; Kozubková, 2005; Kučera, 2008; Pichurov, 2006; Stamp, 2009).

The numerical code is based on a numerical solution of partial differential equations that express the law of conservation of mass (continuity equation), the law of conservation of momentum (Navier-Stokes equations), and the law of conservation of energy (energy equation). The system of equations is solved by a numerical method of finite volumes.

The code contains a wide range of specialized sub-models for different applications. The sub-models can run separately or simultaneously. Both steady state (time-independent) and unsteady (time-dependent) problems at laminar or turbulent fluid flow regimes can be solved.

Many post-processing and visualization techniques and facilities are available, including:

- Values of physical quantities at points, edges, surfaces, or in volumes.
- Two-dimensional graphs of time-dependent or spatial-dependent physical variables.
- Contours of physical variables in two-dimensional cut planes.
- Vectors of physical variables in two-dimensional cut planes.

[1] VŠB - Technical University of Ostrava, Faculty of Safety Engineering, Department of Fire Protection, Ostrava, Czech Republic, ondrej.zavila@vsb.cz

- Iso-surfaces of physical variables.
- Trajectories of flowing particles.
- Animations (figure sequences or clips in defined views, compositions, color schemes, etc.) of courses of modeled time-dependent physical phenomena (considering defined view, space composition, color scheme, etc.).

Calculation run-time depends on the type and complexity of the modeled problem, complexity of the geometry, number of grid cells, number of activated models and sub-models, and capacity of the computer system. The ANSYS Fluent 13.0 was used for modeling the problem discussed in this article (Bojko, 2008; Kozubková, 2008).

Results

Six virtual axial fire ventilators (impeller diameter 1.25 m) were added into the geometry at the roof underside to extract the air from the interior and discharge it to the outside (see Fig. 1 and Fig. 2). All ventilators were placed 27 m above the floor level (3 ventilators on each side of the hall in vertical enclosure walls). The maximum air-flow rate of each ventilator was 100 000 m^3 per hour. The distance between adjacent ventilators in the triplet was 20 m. The simulation assumed activation of all six ventilators.

Fresh air enters the inner space of the arena through all openings (inlets) in the structure (gates and corridors) that are expected to be used as emergency exits and escape routes, respectively (see Tab. 1 and Fig. 5).

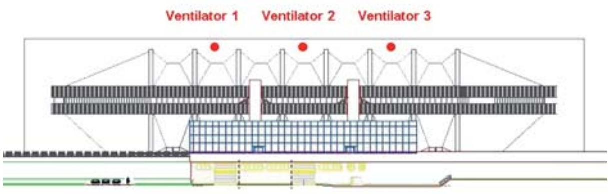

Fig. 1 Fire Ventilators in the Vertical Enclosure Wall of the Indoor Sports Arena - view from the west

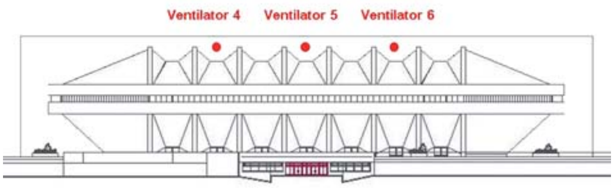

Fig. 2 Fire Ventilators in the Vertical Enclose Wall of the Indoor Sports Arena - view from the east

The geometry represents the three-dimensional shape of the sports arena inner space including the ice resurfacer tunnel. The sports arena has an octagonal ground plan and a total clear height of about 30 m. The greatest width (about 110 m) is at the height of 15 m. Except for small details, the inner space of the arena is vertically and longitudinally symmetric against axes that cross in the center of the floor. The three-dimensional geometry of the arena was created with the DesignModeler software. The geometry was then covered with a grid system of polyhedral type consisting of 550 752 three-dimensional grid cells by using the ANSYS Meshing software (see Fig. 3 and Fig. 4).

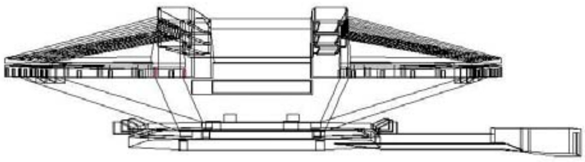

Fig. 3 Geometry of the Sports Arena - view from the north

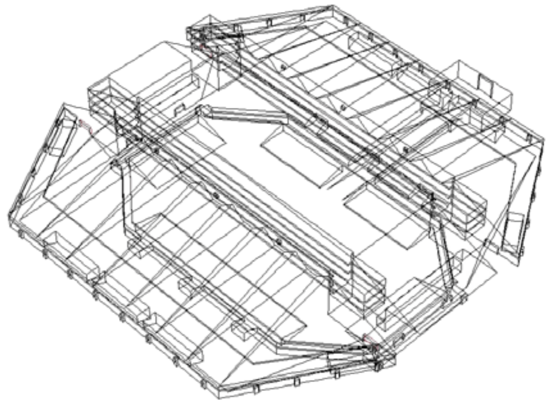

Fig. 4 Geometry of the Sports Arena - view from the northeast (axonometric projection)

The following settings were used for the numerical simulation of the problem. RNG k-ε model of turbulence was chosen for the air-flow field calculation (Bojko, 2008; Kozubková, 2008).

Operational conditions of the model are information (data) that express basic physical characteristics inside the geometry. They were set to:

Atmospheric pressure:	101 325 Pa,
Temperature:	288.15 K (15 ºC),
Gravity acceleration:	9.81 m.s^{-2},
Air density:	1.225 kg.m^{-3}.

Boundary conditions of the model are information (data) that express physical bases of the modeled phenomenon. They are set at geometry

Safety Engineering Series p. 44 - 48, DOI 10.2478/v10281-012-0004-y

boundaries and remain unchanged during the calculation:

Fire ventilators: boundary condition: “velocity-inlet” → settings:

Air-flow velocity: - 22.6 m.s^{-1} (air suction)

Hydraulic diameter: 1.25 m

Intensity of turbulence: 20 %

Inlets: boundary condition: ”pressure-outlet“ → settings:

Hydrostatic pressure: 0 Pa (calculated automatically)

Hydraulic diameter: 1 - 5 m (according to inlet sizes)

Intensity of turbulence: 1 %

Walls: boundary condition: ”wall“ → settings:

Roughness: 0.5 m

Tab. 1 Number and Area of Air-Supply Inlets

Name	Number	Elevation	Inlet Dimensions	Inlet Area	Total Area
Ice resurfacer tunnel (TR)	1	- 2.54 m	4910 x 5000 mm	24.55 m^2	24.55 m^2
Gates from arena (DC)	4	+ 0.0 m	2350 x 2400 mm	5.64 m^2	22.6 m^2
Corridors to arena (PC)	5	+ 2.75 m	2200 x 4000 mm	8.80 m^2	44.0 m^2
Gates to terrace (DT)	8	+ 4.1 m	2150 x 1400 mm	3.10 m^2	24.8 m^2
Corridors to stairways (PS)	4	+ 15.2 m	2350 x 3900 mm	9.17 m^2	36.6 m^2
Gates to outer terrace (DO)	18	+ 15.2 m	1090 x 1970 mm	2.15 m^2	38.7 m^2
TOTAL number of inlets	**40**	**-**	**-**	**-**	**191.3 m^2**

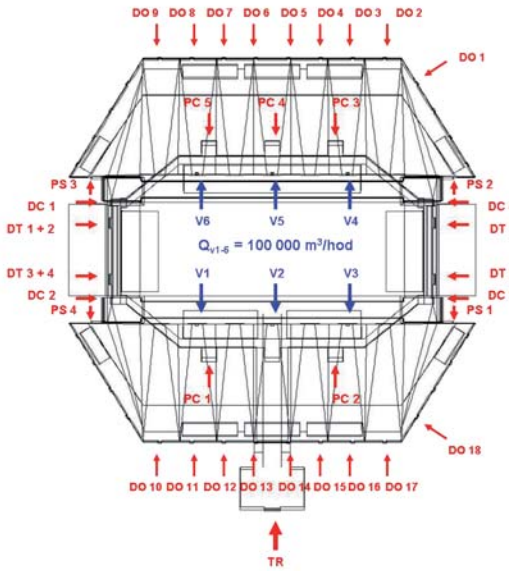

Fig. 5 Placement of Air Inlets, Air Outlets and Ventilators

For better understanding of the modeled situation, see Fig. 5 and Tab. 1. All inlets (gates and corridors) in the structure are labeled with abbreviations for easy identification (see Fig. 5). The problem was calculated as steady-state (time-independent).

The results of the numerical simulation were interpreted by:

a) Values of face-averaged air-flow velocities at each inlet.

b) Filled contours of air-flow velocities in the entire inner space of the arena.

c) Vectors of air-flow velocity fields in the entire inner space of the arena.

The averaged air-flow velocities in the openings (inlets) ranged between 0.72 and 1.18 m.s^{-1} complying with the CEN/TR 12101-5 requirements.

The filled contours of the air-flow field (in two-dimensional cut planes of the three-dimensional geometry) show areas, where the air-flow velocity increases and decreases due to the narrowing of the space (see Fig. 6 to Fig. 8).

The vector fields of the air-flow velocities (in two-dimensional cut planes) show the air-flow directions upon ventilator activation and areas of formation of major turbulent structures (see Fig. 9 to Fig. 11). The figures show that the air travels from the bottom inlets toward the center of the floor where it is drawn up to the ceiling. Under the ceiling, the air is divided into six streams driven to the ventilators and discharged from the building.

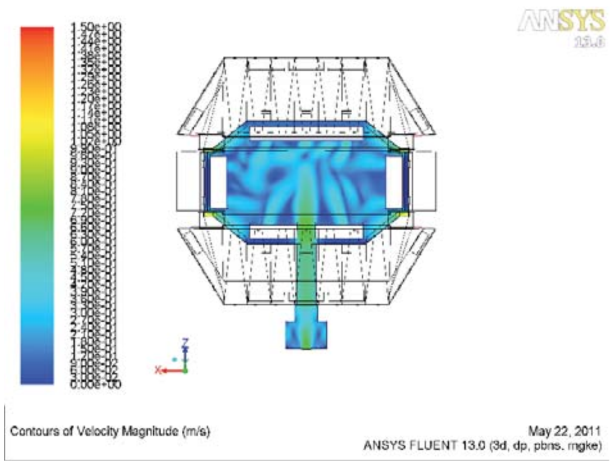

Fig. 6 Filled Contours of the Air-Flow Velocities - horizontal plane cut (1 m above the ground)

Vol. VII, No. 1, 2012

p. 44 - 48, DOI 10.2478/v10281-012-0004-y

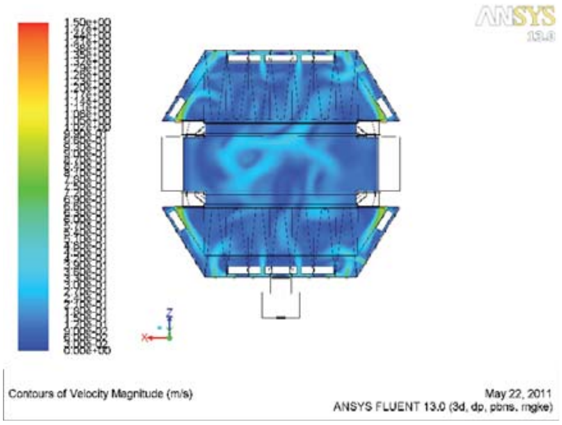

Fig. 7 Filled Contours of the Air-Flow Velocities - horizontal plane cut (16 m above the ground)

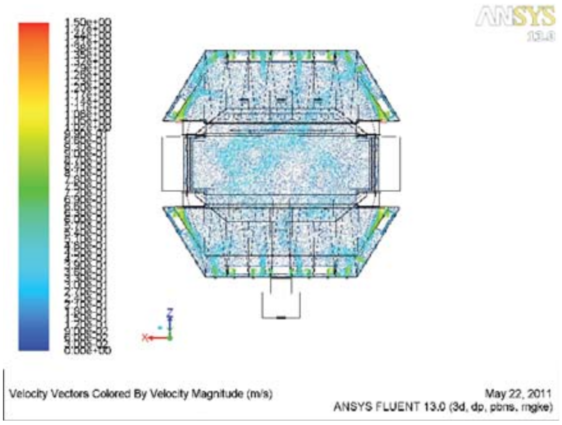

Fig. 10 Vectors of the Air-Flow Velocities - horizontal plane cut (16 m above the ground)

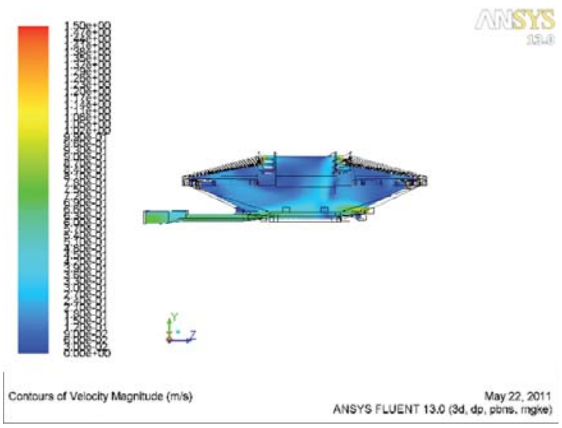

Fig. 8 Filled Contours of the Air-Flow Velocities - vertical crosswise plane cut through the center of the arena

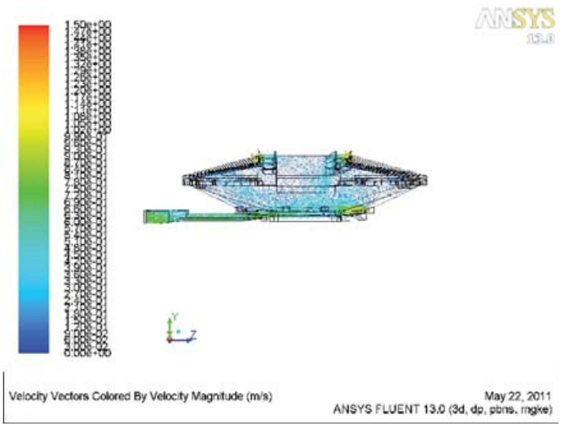

Fig. 11 Vectors of the Air-Flow Velocities - vertical crosswise plane cut through the center of the arena

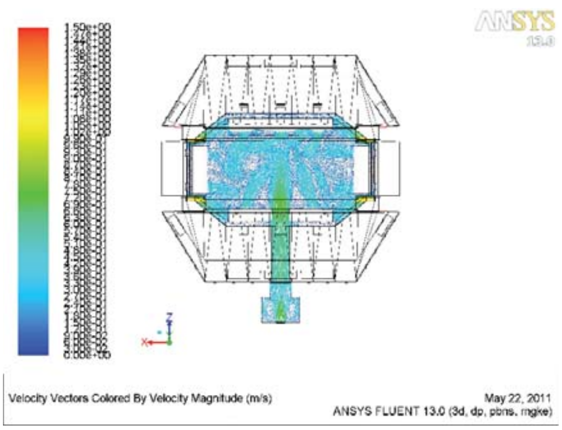

Fig. 9 Vectors of the Air-Flow Velocities - horizontal plane cut (1 m above the ground)

Conclusion

The designed HVAC system in the multifunctional indoor sports arena consisting of six ventilators of the volume flow rate of 100 000 m^3 per hour should ensure one air exchange in 18 minutes. The maximum air-flow velocity of 5 m.s^{-1}, as prescribed by the CEN/TR 12101-5 standard, is not exceeded at any air inlet (gate or corridor).

The numerical simulation was conducted to model only clean air flow field irrespective of heat or smoke aspects. Real fire conditions cannot be predicted with certainty; therefore, they were not included in the simulation. On the other hand, potential hot smoke motion can be predicted from clean air flow field simulation.

In case of fire with smoke production and gas temperature rise in the indoor arena, a change of fluid flow ventilation regime in accordance with physical laws of fluid mechanics (equation of state, Archimedes' principle, etc.) is assumed (Noskievič et al., 1987). Vertical gas flow motion would accelerate in the direction toward the ceiling where the heat and smoke would accumulate. Then the heat and smoke from the roof underside would be discharged to the outside through the fire ventilators. Nevertheless, the hot smoke of a lower density changes the air-flow around the blades of the impellers and would decrease the capacity (volume flow rate) of the fire ventilators. The decreased capacity of the ventilators would increase the air exchange time. Considering that from the past experience only low-heat fires are expected to occur and considering the large volume of the arena, the influence of the change in thermal characteristics on the capacity of the ventilators is expected to be near negligible.

The numerical model contained several simplifications and uncertainties arising from the current technical capabilities of computer technology. For example, beams and complex roof supporting structures, which could cause local turbulence and aerodynamic drag, were not incorporated into the model geometry. To include these details in the macroscale of the whole structure is beyond the possibilities of currently available computer equipment and software. Turbulence and aerodynamic drag at the roof underside would have probably decreased the air-flow velocity at the rest of the arena inner space. Considering the volume of the arena and the capacity of the ventilators, the decrease in the air-flow velocity would be about 0.1 - 0.01 $m.s^{-1}$. Leaks in the structure were not reflected either because their extent could not be reliably determined. Leaks in the structure close to the ventilators could particularly influence the rate of heat and smoke removal from the fire. From a functional point of view, it is therefore preferable to provide ventilators with a greater capacity as it can be easily decreased.

The results of the numerical simulation show that the designed HVAC system should ensure effective fire ventilation of the inner space of the multifunctional indoor sports arena. It should ensure the extraction of combustion products from the escape routes and the supply of fresh air, thus enabling safe emergency evacuation and fire suppression activities. The air-flow velocities at the emergency exits and escape routes do not pose a risk or danger for the evacuees.

Acknowledgments

Author acknowledges the financial support of the SPII 1a10 45/07 project of the Ministry of the Environment of the Czech Republic.

References

BOJKO, Marian (2008). *Návody do cvičení „Modelování proudění“ - FLUENT*, 1st. Ostrava, Czech Republic: VŠB - TU Ostrava, 2008, 141 p. ISBN 978-80-248-1909-9 (in Czech).

CEN/TR 12101-5:2005. *Smoke and Heat Control Systems - Part 5: Guidelines on Functional Recommendations and Calculation Methods for Smoke and Heat Exhaust Ventilation Systems*, 1st edit. Praha, Czech Republic: ČNI, 2008, 100 p.

KOZUBKOVÁ, Milada (2008). *Modelování proudění tekutin, FLUENT, CFX*, 1st. Ostrava, Czech Republic: VSB - TU Ostrava, 2008, 153 p. (in Czech).

KOZUBKOVÁ, Milada, DRÁBKOVÁ, Silva (2005). Wind tunnel simulation and their importance for the numerical modelling of atmospheric flow and pollutant dispersion. In: *Transactions of VSB - Technical University of Ostrava, Safety Engineering Series*. Ostrava, Czech Republic: VŠB - Technical University Ostrava, 2005, pp. 25-36. ISBN 80-248-0940-0 (in Czech).

KUČERA, Petr, MIKLÓS, Pavel (2008). Fundamentals of verification of mathematical fire models. In: *Transactions of VSB - Technical University of Ostrava, Safety Engineering Series*. Ostrava, Czech Republic: VSB - Technical University Ostrava, 2008, pp. 47-56. ISBN 978-80-248-1920-4 (in Czech).

NOSKIEVIČ, Jaromír et al. (1987). *Mechanika Tekutin*. 1st. Praha, Czech Republic: Státní nakladatelství technické literatury, 1987. 354 p. (in Czech).

PICHUROV, G., STANKOV, P., MARKOV, D. (2006). HVAC control based on CFD analysis of room airflow. *IFAC Proceedings Volumes*. 2006, Vol. 1, No. 1, pp. 213-218. ISSN 14746670.

STAMP, B.J., MAXSON, S.T. (2009). Simulation driven HVAC design. *Engineering Systems*. 2009, Vol. 26, No. 1, pp. 64-74. ISSN 08919976.