International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 10 Oct 2021, pp: 412-419 www.ijaem.net ISSN: 2395-5252

Review of Building Energy Performance by CFD Analysis

(1)Aruna Mohan, (2)A.Madelin Madona
(1)Assistant Professor, Thiagarajar College of Engineering, Madurai 625 015
ORCID: 0000-0001-9009-6672
(2)Bachelor of Engineering Student, Thiagarajar College of Engineering, Madurai 625 015

Submitted: 01-10-2021 Revised: 10-10-2021 Accepted: 12-10-2021

ABSTRACT
Building sector is the highest contributor of carbon dioxide emissions. Two types of energy a prevalently used in building sector, they are embodied energy and operation energy. Embodied energy contributes to about 20% of total energy and operational energy contributes to about 80% of total energy. Building materials has large effects on heat radiation and thermal comfort of the building. Orientation of the building plays pivotal role in heat dissipation. Out of three heat transfer mechanisms, convection, conduction and radiation, radiation effect has large potential in energy savings through building orientation, materials used in the construction, applying green building standards. Though it has largest potential, calculation of radiation effect is a difficult task. Computational Fluid Dynamics is one of the tool which helps in computing the heat radiation effects in the building. It involves a grid system and set of mathematical equations are applied to solve the complex problems.

Key Words: Radiation, CFD, Built Environment, Heat Flux, Thermal Comfort

INTRODUCTION:

In the previous decades cities have been continually increasing. Around the planet, there are billions upon billions of structures. The inhabitants anticipate a well-designed building with adequate thermal comfort. Without the use of any mechanical devices, a good house design preserves the inside atmosphere favourable and pleasant throughout the majority of the year. As a result, air circulation must be one of the most significant factors to consider while designing a structure. Health, comfort and energy consumption are essential criteria to consider in connection to design of building. The term "thermal comfort" refers to a person's satisfaction with their thermal surroundings. It refers to a set of circumstances in which the majority of individuals are at ease. Thermal comfort is one of the most significant factors in enhancing occupant comfort and happiness with their interior environment. Air temperature (Ta), air velocity, mean radiant temperature (MRT), water vapour pressure (Pa) or relative humidity (RH), relative air velocity (Vt), and human characteristics such as sickness, clothing, and metabolic heat all influence thermal comfort. Problems involving heat transfer that occur in the design of buildings and building energy systems include reducing energy use in buildings, producing satisfactory thermal comfort conditions within buildings, providing natural ventilation in buildings, and minimising the spread of pollutants within a building. As electronic components get smaller and smaller, the amount of space available for heat dissipation shrinks.

Bahlaoui at al[1] have studied the relationship between radiation and natural convection in a tall rectangular inclined cavity(θ =45°) and a vertical cavity (θ =90°) .A Numerical method is studied.The numerical method adopts the Finite Difference procedure,Alternate Direction Implicit Method and the Gauss Seidel Method. The Rayleigh number and the emissivity of the cavity walls are important parameters. The cavity wall is heatedon one side(constant temperature)and cooled on the other side of the wall (uniform temperature).The properties of Fluid obeys Boussinesq approximation. External Ventilation is considered to boost the cooling process. Radiosity method is employed to calculate the heat transfer between the walls.

Venkateshan at el[2] have conducted natural convection heat transfer experiments on discrete heat sources protruding at different positions on the substrate and studiedthe influence of surface radiation heat transfer on the optimal distribution of discrete heat sources under natural convection.It can reduce the temperature by as much 12%.Taking into account the influence of surface radiation heat transfer ,an empirical correlation is formed between dimensionless steady state temperature (θ) and dimensionless geometric distance (τ) parameters, which is suitable for all possible configurations of heat sources on the substrate.The Experimental set up uses Bakelite material to provide low thermal conductivity,80/20 Nichrome wire is used as heating element and a wooden cavity is employed to prevent external disturbances. Aluminiumis considered to be isothermal.

Gilles Desrayaud et al [3] explored steady,two dimensional laminar natural convection during a very parallel vertical channel, with one protruding heat module using numerical simulations. Conduction is a key cooling process in heat transfer analysis that must be taken into account.

Chen and Liu [4] ,and Chen et al[5] , have done experimental studies to cool down the electronic package by seeking out the optimum spacing along the heated elements. An advanced thermal performance can be obtained, if the centre to centre distance between the heat sources follow the geometric series.

Saravanan et al [6] theoretically studied to know the interaction between surface radiation and natural convection in an air filled cavity cantered on a central heating plate positioned horizontally and vertically.It indicated a more robust homogenization of temperature field within the cavity by radiation. Liu and Phan-Thien [7] numerically studiedthe problems of conjugate radiation,heat conduction and convection during the differential heating of a heated block during a vertical differentially heated square cavity. They concluded that radiation encompasses a strong influence on temperature and velocity distribution, so when more heat is generated in the block,emissivity has a great influence on global flux.Sigel and Howell [8] used net radiation method to determine radiative flux.Patankar [9] uses the SIM –PLE algorithm. The tri-diagonal matrix algorithm (TDMA) is employed to solve discrete equations. Hottel and Saroffim [10] used cross string method to observe thefactors between the surface elements.

Behnia et al [11],studied combined natural convection and radiation in a rectangular, 2D cavity containing a non-participating fluid by numerical method.One wall of the cavity is isothermal,opposite wall is partially transparent and other two walls are adiabatic.Within the absence of direct isolation,results are obtained for a square cavity an ambient temperature of 20° C,a vertical hot wall at 150° C and $10^4 \leq Ra \leq 3\times10^5$.

Agung Murti Nugroho et al [12], studied the thermal comfort in Malaysia's single stored terraced Houses for under annual climatic conditions and discussed the validation of computational Fluid Dynamicsresults. The CFD technique and field measurement are employed to determine the thermal environment and comfort conditions.Use Compact Humidity Logger (CHL) and Thermal Data Logger (TDG) for field measurements.Relative Humidity and air temperature are the foremost important determinants of thermal comfort.Outdoor climatic conditions,Indoor Climatic Condition, Environmental parameters (relative humidity,wind speed,air temperature and mean radiant temperature) are all recorded by Globe Weather Station, Thermal &compact data Logger and Inova thermal Data Logger respectively. Air temperature and relative humidity are measured using a DicksonCompact Humidity Loggers.The results of CFD and Field measurements are slightly different.The Auliciems' neutrality air temperature equation is employed to define the comfort air temperature of Malaysians and bioclaimatic Chart was proposed.The Ventilation can be equipped with a Solar Chimney. In past ten years, solar chimneys have received a lot of attention in numerous investigations(Barrozi, 1992; Bouchair, 1994; Gan, 1997; Hirunlabh, 1999; Khedari, 2000; Alfonso, 2003; Waewsak, 2003; Ong, 2003; Drori, 2004; Satwiko, 2005 and Bansal, 2005).Solar Chimney : Increase Indoor air velocity and reduce Indoor air Temperature. The comparison of field measurements and CFD simulationshows that the results are in good agreement with field measurement.Thermal CFD models indicated that higher speed are needed to achieve thermal comfort. The results show that the construction of single storey terraced houses is ineffective in providing natural ventilation for achieving thermal comfort.

Jonas Allegrini et al [13],analysed the heat flow in Urban regions.The urban microclimate is very different from the rural microclimate.Due to the Urban Heat Island effect (UHI),the temperature in Urban areas are generallyhigher. The UHI effect is mainly caused by reduction of sky observation factors, materials with high heat capacity, Human and animal made heat, noevaporation and reduction turbulent convection. Most of the heat flow in

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 10 Oct 2021, pp: 412-419 www.ijaem.net ISSN: 2395-5252

urban areas is caused by the absorption of solar radiation on the surface and ground of buildings.The BES model is used to determine the surface temperatures ofthe buildings and the ground in the urban areas.The BES model considers short wave and longwaveradiation,surface convection heat transfer,and heat conduction through the walls andground. In order to predict the temperature and flow field, a CFD simulation was performed,using the surface temperature determined byBES as the boundary condition.Two types of research are involved: Field Measurement and Numerical Simulation.The advantage of Numerical Simulations is that it is easy to conclude and provides high resolution results.The disadvantage of Field measurement are high cost,complexity and difficult to obtaining the results. Heat fluxes for six different urban morphologies were studied with CFD (computational fluiddynamics) and building energy simulations (BES).External shading device are used to protect building from solar radiation. For the near-wall modelling, wall functions (Launder and Spalding 1974) [14] with no-slip boundary circumstancehad been used as a compromise among accuracy and computational cost. The feasible overestimation of the wall heat flux on theconstructing façades due to the use of wall functions (Allegrini et al. 2012) [15] is believed to be comparable for all studied cases.For the BES CitySim (Kämpf 2009)[16] is used. CitySim is a simulation tool that models the energyfluxes in every city, with a size starting from a little neighbourhood to a complete city. In CitySim,complex radiation models for solar and long-wave radiation are used,which canexplain the radiation exchange between neighbouring buildings, the ground and the environment.

The Perez AllWeather (Perez et al. 1993)[17] and the Simple Radiosity algorithmic program (Robinson and Stone, 2006)[18] areaccustomed work out hourly irradiations of short and longwave radiation on building surfaces. Multipleiterations for the radiation calculations are executed to attain steady results. The heat flowthrough the walls is set with a model based on the analogy with associate electrical circuit. In order to study the urban microclimate, astable RANS (ReynoldsAveraged NavierStokes) 3D CFD simulation with a turbulence model kε was performed using OpenFOAM. To account for buoyancythe Boussinesq approximation is used. The second-orderscanning scheme and SIMPLE algorithm is used for pressure and velocity coupling. Pressure interpolation is of second-order. Discussed the resultofa slightlyhigheramount of control thanin urban areas. Twodifferent testvolumes.The method is defined as:higher control volume and lower control volume. The results show that almost of the heatescapesfromupstream or downstream in urban areas.

Haghighat et al [19] ,studied the distributions of Indoor air velocity,Air temperature, pollutant Concentration and Ventilation effectiveness in a two zone enclosure in orderto develop a three dimensional numerical model(a simple algorithmic rule and a turbulence model consisting of two equation k-ε). The false-time step andADI iteration method are used. The processes of air and contaminant flow through a building are three-dimensional and take place under complex conditions. The processes can be described by a set of conservation equations namely the continuity, momentum, and energy equations with applicable boundary conditions. Since these nonlinear, second-order partial differential equations are in addition to one another, aprecise solution has not yet been obtained. Several researchers have looked into natural convection in an enclosed cavity (i.e., Catton, 1978; Ostrach, 1982;Markatos and Pericleous, 1984; Lin and Nansteel, 1987; Hadjisophocleouset al., 1988; Gadgil et al., 1984).Experimental measurements and numerical simulation are generally used to investigate the intra and interzone convective heat and mass transfer in buildings.

The air movement in an exceedingly multizone enclosure is tormented by many parameters, such as the dimension of the enclosure, the size and the location of the door opening, the locations of the ventilation inlet and outlet, the temperature conditions of walls,etc.A numerical model provides overall view of field distribution. Natural convection in an enclosure with both laminar and turbulent flows was explored by Markatos and Pericleous [20].The two-dimensional laminar flow was solved in the form of finite difference stream functioning equations and modified Taylor series approximations of the vorticity equations.The SIMPLEST algorithmic rule was adopted to resolve the finite difference equations. Chen et al. (1990 a) [21]utilised a low Reynolds number k-e turbulence model to predict velocity and temperature distributions in an enclosure with natural convection flow. They found that the low Reynolds number k-e model gives a more robust velocity profile near the walls and more accurate simulations of convective heat transfer from walls to room air than the k-e model for calculating indoor air movement. Chen et al.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 10 Oct 2021, pp: 412-419 www.ijaem.net ISSN: 2395-5252

(1990b)[22] investigated indoor air quality and thermal comfort in a ventilated enclosure quantitatively for six scenarios employing various types of air diffusers. Blockage by furniture and pollutants emitted by furniture were stimulated, again the Reynolds model with the low numberk-e was used. Berne and Villand (1987)[23] described the three-dimensionalthermo hydraulic code TRIO-VF which can handle laminar or turbulent flows or incompressible fluids with the Boussinesq approximation. This code was applied to model individual ventilated enclosures with a complex environment. The airflow patterns were demonstrated in the ventilated enclosures under different boundary conditions. The PHOENICS code has been used by various researchers for two or three-dimensional analysis of convective heat and mass transfer inbuildings(i.e., Holmes, 1982; Markatos,1983; Jones and Sullivan, 1985; Chen and Van der Kooi,1988). Markatos and Cox (1984)[20] applied the modified PHOENICS to predict the development of a fire and the distribution of the concentration of the pollutants within a shopping mall. To calculate the air movement and heat transfer in a room heated by a radiator, Lemarie (1987)[24] used the CHAMPHxN computer code (Pun and Spalding, 1976) [25]in conjunction with a radiation model.The door opening velocity distributionspredicted by Concordia and the PHOENICS code are good.The wall function method is the most widely method of treating this problem.The wall functions are based on the one-dimensional steadystate boundary layer equations and the mixing-length hypothesis.

The wall function technique works on the idea of using the momentum flux due to shear stress and the heat flux at solid surfaces to alter the source terms in the conservation equations for grid nodes near solid surfaces.The aim of grid dependence test is to select a mesh system with the smallest possible grid number and satisfactory accuracy.Computations are performed with an increasing number of grid nodes in the grid dependence test until further increment shows negligible changein solution. With natural convection,it has beenfound that the height of the door has a strong influence on the heat convection from the hot zone to the cold zone, while the door opening is shifted from the centre of the partition towards a side wall slightly increases the convection heat transfer (Haghighat et al., 1989)[26].TheModel was validated by comparing the predicted results with available experimental data. The results obtained show that proper location of supply,exhaust, and door openings can be important in attempting in order to improve indoor air quality in each zone.

Cinzia Buratti et al[27],CFD Analysis and experimental validation were used to develop and optimize a new ventilated brick wall.The prototype was created. In the past, ventilated walls were used to protect structures from moisture and deterioration. CFD simulations were also used to look into the ventilation into the air gap. By analysing the velocity profile into the air gap, the verified CFD model was utilised to optimise the ventilation apertures. The use of ventilated walls is popular in construction, including internal double walls (Fraisse et al)[28] in some circumstances, and notably in new structures, although their installation necessitates careful planning. They are therefore extensively researched, not only in terms of energy efficiency, but also in terms of cost-benefit and environmental considerations (Pulseli et al)[29].Several researches [30-33] looked at a different method of obtaining ventilation in hollow core slabs and block walls by establishing numerical techniques that were validated using experimental data.

The effects of incorporating micro-encapsulated phase change materials (mPCM) for cooling of office buildings were evaluated both numerically and experimentally in [30], and the results showed that the cooling potential improves with different phase changing temperatures, depending on the wall density. In the case of hollow block ventilated walls [31–33], many models were created using various techniques, including a semi-dynamic heat transfer model [33], a Frequency-Domain Finite Difference FDFD model and a CFD model [31], and a Dynamic Thermal Network Model [32]. Yu et al. [33] utilised the semi-dynamic heat transfer model to estimate the thermal behaviour of a hollow ventilated block with exhaust air from the HVAC system running through the cavity. The model was able to estimate the hollow block's surface temperatures as well as the heat flux reduced due to cavity ventilation. Using a model based on thermal and aerodynamic balancing equations, Saadon et al. [34] analysed the characteristics defining a partly transparent ventilated PV façade integrated in the envelope of a building. The model's outputs (surface and air temperatures, mass flow rates, and PV power output) were used to calculate the heating and cooling demands of a building with a PV ventilated façade in different climate conditions in France using the TRNSYS code.

Diarce et al[35]. Utilised the Fluent CFD model to predict the thermal behaviour of a innovative ventilated wall made of PCM materials.

By simulating different wind conditions in Catania, Gagliano et al. [36] were able to anticipate and assess the behaviour of vented façades in the summer (Italy). Inside the air gap of the façade, the temperature and air velocity were estimated, and the various effects of wind forces were examined. The new ventilated wall should have the following features: the external layer should be built with 12 cm thick face bricks; the external layer should be completely supported by a steel anchorage system for structural reasons; and only a few ventilation openings should be present along the width of the wall to ensure structural stability. The 3D ventilated model was then utilised to optimise the openings on the wall's configuration. Six alternative designs were studied, each with a different distribution of apertures, and the optimum option for air gap ventilation was chosen.

Patrick H.Oosthuizen et al[37],studied the application of CFD in the analysis of heat transfer problems in building energy. Computational fluid dynamics (CFD) and Energy Simulation (ES) computer programs has been more popular in recent years for the design and assessment of building energy systems. CFD, on the other hand, requires inputs from ES, such as heating/cooling load and, in some circumstances, wall surface temperatures. As a result, combining ES with CFD is a highly appealing strategy. A designer may create an energy-efficient, thermally pleasant, and healthy building using information from both ES and CFD simulations. The atrium building is an example of a building typology where CFD analysis has shown to be especially beneficial. The following are some of the reasons why atria are currently so popular: They substantially enhance the building's attractiveness.They allow sunshine to penetrate deep into the building and as a result boost the physical and psychological well-being of the building's residents leading to an improvement in morale.

They have the ability to drastically cut the building's energy use. The Boussinesq Approximation is employed. Smoke is frequently the leading cause of mortality in the event of an interior fire. One of the major goals of any fire prevention system's design is to protect the building's inhabitants during a fire.

In order to better understand how smoke spreads in huge structures, numerical models are used. The findings demonstrating the basic aspects of the spread of smoke and calculated spreads of smoke in atria have been found to be in excellent agreement with the actual data [56]- [62]. CFD models have been successfully applied to simulate fires in buildings of various forms. Because people spend more time indoors than outside and because contamination levels are higher in indoor circumstances than in outdoor ones, the spread of pollutants within buildings is typically of more concern. CFD has proved to be a useful tool for simulating the movement of pollutants in buildings. CFD techniques may be used to model both interior and outdoor pollutant dispersal. Natural ventilation of buildings is ventilation caused by buoyant forces generated by the greater average temperature of the inside air compared to the outside air temperature, or by pressure differences caused by wind-flow over the building. To provide pleasant circumstances for the occupants, the supply air conditions may be adjusted using CFD. [64] – [100] cover various elements of both natural and hybrid ventilation. CFD projections were compared to certain existing observations of flow and temperature distribution in atria. These comparisons show that CFD techniques can accurately anticipate atria conditions.

COMPUTATIONAL FLUID DYNAMICS:

Computational Fluid Dynamics (CFD) is a method for modelling, predicting, and visualising how fluids, such as gas or liquid, move using techniques from physics, applied mathematics, and computer science. [1]. CFD may be used to solve issues such as turbomachinery, building performance under various weather situations, storm analysis, heat exchangers, fully developed turbulent flow in pipes, fluid flow and air foil analysis in aeroplanes, and forced convection cooling of electronic chips in processors. Many academics have utilised computational fluid dynamics to analyse the interior environment since it is a strong tool.

CFD IN BUILDING DESIGN:[37]

- By forecasting the distributions of air velocity, temperature, moisture, turbulence intensity, and pollutant concentration around structures, CFD can assist optimize building sites. Good site planning may successfully shield buildings from the negative impacts of pollutants released by nearby activities, as well as improve outdoor pedestrian comfort and improve building energy efficiency.
- The possibility of employing buoyancy-driven natural ventilation generated by human and machine heat creation within the building, as well as solar energy absorption within the building.
- The efficiency of heat transfer processes within the building and at building surfaces, such as heat transfer via windows and the influence of

window coverings on this heat transfer, as well as the effect of vent discharges on the flow over the windows and therefore on heat transfer through the windows.

- The spread of smoke caused by a fire in the structure.

APPLICATIONS OF CFD:

- ✓ The drying process of air currents inthe areas surrounding a buildinghas also been studied[2], leading to the development of an equation that can estimate moisture transport by convection;The simple equation was tested using experimental data showing that it can reliably predict moisture transfer.
- ✓ CFD was used by An-Shik Yang et al.[3] in urban and community planning to simulate, acquire flow parameters and features around various structures in order to provide ventilation. The simulation domain was set to 3kmx2kmx0.6km. The turbulence model was a conventional k- two equation model.
- ✓ CFD applications are not restricted to the aforementioned sectors, and as a result, they are widely used in a variety of disciplines of research.[4]
- ✓ Weather forecasting and disaster risk assessment.[4]

ADVANTAGES:

- o CFD is also a cost-effective method since it converts real fluids into digital images, allowing for a more thorough, longer, and uncorrupted study. This is demonstrated by improvements in different fields and the resulting cost reductions. CFD may also be used to generate controlled simulations that aren't based on real-world scenarios. Consider a nuclear explosion or a big volcanic eruption. [6]
- o When compared to physical modelling approaches, CFD produces faster results for complicated modelling geometries [5]. Physical modelling techniques consume more time, space, and money.

DISADVANTAGE:

Although there are several benefits to utilising CFD, there are also drawbacks to the use. It can, for example, disagree from various physical model findings, resulting in a deadlock as to which should be considered the more legitimate result [5].

REFERENCES:

[1]. A. Bahlaoui ,A. Raji,M. Hasnaoui, (2006),"Combined effect of radiation and natural convection in a rectangular enclosure discreetly heated from one side", International Journal of Numerical Methods for Heat & Fluid Flow, Vol. 16 Iss 4 pp. 431 - 450

[2]. Tapano Kumar Hotta ,Pullarao Muvvala ,S. P. Venkateshan , “Effect of surface radiation heat transfer on the optimal distribution of discrete heat sources under natural convection” Heat Mass Transfer (2013) 49:207–217

[3]. Desrayaud G, Fichera A, Lauriat G (2007) “Natural convection air-cooling of a substrate-mounted protruding heat source in a stack of parallel boards”. Int J Heat Fluid Flow 28(3):469–482

[4]. Chen S, Liu Y (2002) “An optimum spacing problem for three-by-three heated elements mounted on a substrate”. Heat Mass Transf39(1):3–9

[5]. Chen S, Liu Y, Chan S, Leung C, Chan T (2001)” Experimental study of optimum spacing problem in the cooling of simulated electronic package”. Heat Mass Transf 37(2):251–257

[6]. S. Saravanan , C. Sivaraj “ Coupled thermal radiation and natural convection heat transfer in a cavity with a heated plate inside” International Journal of Heat and Fluid Flow 40 (2013) 54–64

[7]. Liu, Y., Phan-Thien, N., 1999. “A complete conjugate conduction convection and radiation problem for a heated block in a vertical differentially heated square enclosure”. Comput. Mech. 24, 175–186.

[8]. Siegel, R., Howell, J.R., 2002. Thermal Radiation Heat Transfer, fourth ed. Taylor and Francis Group, New York.

[9]. Patankar, S.V., 1980. “Numerical Heat Transfer and Fluid Flow”. Hemisphere Publishing Corporation, Taylor and Francis Group, New York. Ramesh, N., Venkateshan, S.P., 1999.” Effect of surface radiation on natural convection in a square enclosure”. J. Thermophys. Heat Transfer 13 (3), 299–301.

[10]. Hottel, H.C.,Saroffim, A.F., 1980.” Radiative Heat Transfer”. Mcgraw Hill, New York

[11]. M. BEHNIA, J. A. “Reizes and G. De Vahl Davis Combined Radiation and Natural Convection In A Rectangular Cavity With A Transparent Wall And Containing A Non-Participating Fluid” International Journal for Numerical Methods In Fluids, Vol. 10, 305-325 (1990)

[12]. Agung Murti Nugroho, Mohd Hamdan Ahmad and Dilshan Remaz Ossen “A Preliminary Study of Thermal Comfort in Malaysia's Single Storey Terraced Houses”
[13]. Allegrini, J., Dorer, V., and Carmeliet, J.,Coupled “CFD, radiation and building energy model for studying heat fluxes in an urbanEnvironment with generic building configurations, Sustainable Cities and Society”(2015)
[14]. Launder, B.E., Spalding, D.B., 1974. The numerical computation of turbulent flows. Computational MethodApplication M Eng 3, 269-289.
[15]. Allegrini, J., Dorer, V., Carmeliet, J., 2012. “An adaptive temperature wall functions for mixed convective flows at exterior surfaces of buildings in street canyons.” Build. Environ. 49, 55-66.
[16]. Kämpf, J., 2009. On the modelling and optimisation of urban energy fluxes, PhD thesis Lausanne.
[17]. Perez, R., Seals, R., Michalsky, J., 1993. All-weather model for sky luminance distribution-preliminaryconfiguration and validation. Sol. Energy 50 (3), 235–243.
[18]. Robinson, D., Stone, A., 2006. Internal illumination prediction based on a simplified radiosity algorithm. Sol. Energy 80 (3), 260-267.
[19]. F. Haghighat,Z. Jiang,J. C. Y. Wang Air Movement in Buildings UsingComputational Fluid Dynamics
[20]. Markatos, N. C, and Pericleous, K. A., 1984, "Laminar and TurbulentNatural Convection in an Enclosed Cavity," International Journal of Heat andMass Transfer, Vol. 27, No. 5, pp. 755-772
[21]. Chen, Q., Moser, A., and Huber, A., 1990, "Prediction of Buoyant, Turbulent Flow by a Low-Reynolds-Number k - e Model," ASHRAE Transactions,Vol. 96, Pt. 1, pp. 564-573.
[22]. Chen, Q., Moser, A., and Suter, P., 1990, "Indoor Air Quality and Thermal Comfort Under Six Kinds of Air Diffusion," presented at ASHRAE Winter Meeting.
[23]. Berne and Villand M., 1987, "Prediction of Air Movement in a Ventilated Enclosure with 3-D Thermohydraulic Code TRIO," ROOMVENT-87, Vol. 3, Stockholm, pp. 1-15.
[24]. Lemaire, A. D., 1987, "The Numerical Simulation of the Air Movement andHeat Transfer in a Heated Room Resp. A Ventilated Atrium, "Proceedings ofthe International Conference on Air Distribution in Ventilated Space, ROOMVENT-87, Stockholm.
[25]. Pun, W. M., and Spalding, D. B., 1976, "A General Computer Program forTwo-Dimensional Elliptic Flows," HTS/76/2, Imperial College, London.
[26]. Haghighat, F., Jiang, Z., and Wang, J. C. Y., 1989, "Natural Convectionand Airflow Pattern in a Partitioned Room with Turbulent Flow," ASHRAETransactions, Vol. 95, Pt. 2, pp. 600-610.
[27]. Cinzia Buratti, Domenico Palladino, Elisa Morett, Rocco Di Palma Development and optimization of a new ventilated brick wall: CFD analysis and experimental validation Energy & Buildings 168 (2018) 284–297
[28]. G. Fraisse , R. Boichot , J.L. Kouyoumji , B. Souyri , Night cooling with a ventilated internal double wall, Energy Build. 42 (2010) 393–400
[29]. R.M. Pulselli , E. Simoncini , N. Marchettini , Energy and emergy based cost–ben- efit evaluation of building envelopes relative to geographical location and cli- mate, Build Environ. 44 (2009) 920–928.
[30]. A. Faheem , G. Ranzi , F. Fiorito , C. Lei , A numerical study on the thermal per- formance of night ventilated hollow core slabs cast with micro-encapsulated PCM concrete, Energy Build. 127 (2016) 892–906.
[31]. J. Yu, J. Yang, C. Xiong, Study of dynamic thermal performance of hollow block ventilated wall, Renew. Energy 84 (2015) 145–151.
[32]. C. Xiong , J. Yu , J. Yang , Study of simplified dynamic thermal network model for the hollow block ventilated wall, Procedia Eng. 121 (2015) 1304–1311 .
[33]. J. Yu , J. Huang , X. Xu , H. Yea , C. Xiong , J. Wang , L. Tian , A semi-dynamic heat transfer model of hollow block ventilated wall for thermal performance pre- diction, Energy Build. 134 (2017) 285–294.
[34]. S. Saadon, L. Gaillard , S. Giroux-Julien , C. Ménézo, Simulation study of a naturally-ventilated building integrated photovoltaic/thermal (BIPV/T) envelope, Renew. Energy 87 (2016) 517–531.
[35]. G. Diarce , Á. Campos-Celador , K. Martin , A. Urresti , A. García-Romero , J.M. Sala , A comparative study of the CFD modeling of a ventilated active façade includ- ing phase change materials, Appl. Energy 126 (2014) 307–317 .

[36]. A. Gagliano , F. Nocera , S. Aneli , Thermodynamic analysis of ventilated façades under different wind conditions in summer period, Energy Build. 122 (2016) 131–139.

[37]. Patrick H.Oosthuizen , Marilyn Lightstone, USE OF CFD IN THE ANALYSIS OF HEAT TRANSFER RELATED PROBLEMS THAT ARISE IN BUILDING ENERGY STUDIES Proceedings of the 14th International Heat Transfer Conference IHTC14 August 8-13, 2010, Washington, DC, USA.

[38]. Hunt, G.R. and Kaye, N.G., 2001, "Virtual Origin Correction for Lazy Turbulent Plumes," Journal of Fluid Mechanics, 435, pp. 377–396.

[39]. Gutiérrez-Montes, C., Sanmiguel-Rojas, E., Kaiser, A.S., Viedma, A., 2008, "Numerical Model and Validation Experiments of Atrium Enclosure Fire in a New Fire Test Facility," Building and Environment, 43, pp. 1912–1928.

[40]. Chow, W.K., Li, S.S., Gao, Y. and Chow, C.L., 2009, "Numerical Studies on Atrium Smoke Movement and Control with Validation by Field Tests," Building and Environment, 44, pp. 1150–1155.

[41]. Qin, T.X., Guo, Y.C., Chan, C.K. and Lin, W.Y., 2009, "Numerical Simulation of the Spread of Smoke in an Atrium under Fire Scenario," Building and Environment, 44, pp. 56–65.

[42]. Andersen, K.T., 2003, "Theory for Natural Ventilation by Thermal Buoyancy in One Zone with Uniform Temperature," Building and Environment, 38, pp.1281–1289.

[43]. Etheridge, D. and M. Sandberg, M., 1996, "Building Ventilation: Theory and Measurement," England: John Wiley & Sons Ltd.

[44]. Y. Li, F. Haghigat, K.T. Andersen, H. Brohus, P.K. Heiselberg, E. Dascalaki, G.V. Fracastoro, and M. Perino, 1999, "Analysis Method for Natural and Hybrid Ventilation: an IEA ECB Annex 35 literature review," Proceeding of the 3rd International Symposium on Heating, Ventilation and Air Conditioning, Sheseng, China, November.

[45]. Guang Xu, Kray D. Luxbacher, SaadRagab, JialinXu, and Xuhan Ding, 2017, Computational fluid dynamics applied to mining engineering: a review, International Journal of Mining, Reclamation and Environment, Vol. 31, Iss. 4, pp. 251-275.

[46]. D. Davidovic , J Srebric , EFP. Burnett, Modeling convective drying of ventilated wall chambers in building enclosures, Int. J. Thermal Sci. 45 (2006) 180 .

[47]. An-Shik Yang, Chih-Yung Wen, Yu-Chou Wu,Yu-Hsuan Juan, and Ying-Ming Su, 'Wind Field Analysis for a High-rise Residential Building Layout in Danhai, Taiwan', Proceedings of the World Congress on Engineering 2013 Vol II, WCE 2013, July 3 - 5, 2013, London, U.K.

[48]. 1Senan Thabet,Thabit H. Thabit Computational Fluid Dynamics: Science of the Future International Journal of Research and Engineering

[49]. Linfield, K. W., and Mudry, R. G., 2008, Pros and cons of CFD and physical flow modeling.

[50]. Tu, J., Yeoh, G., and Liu, C., 2012, Computational Fluid Dynamics – A practical Approach, 2nd Edition, Elsevier, UK.