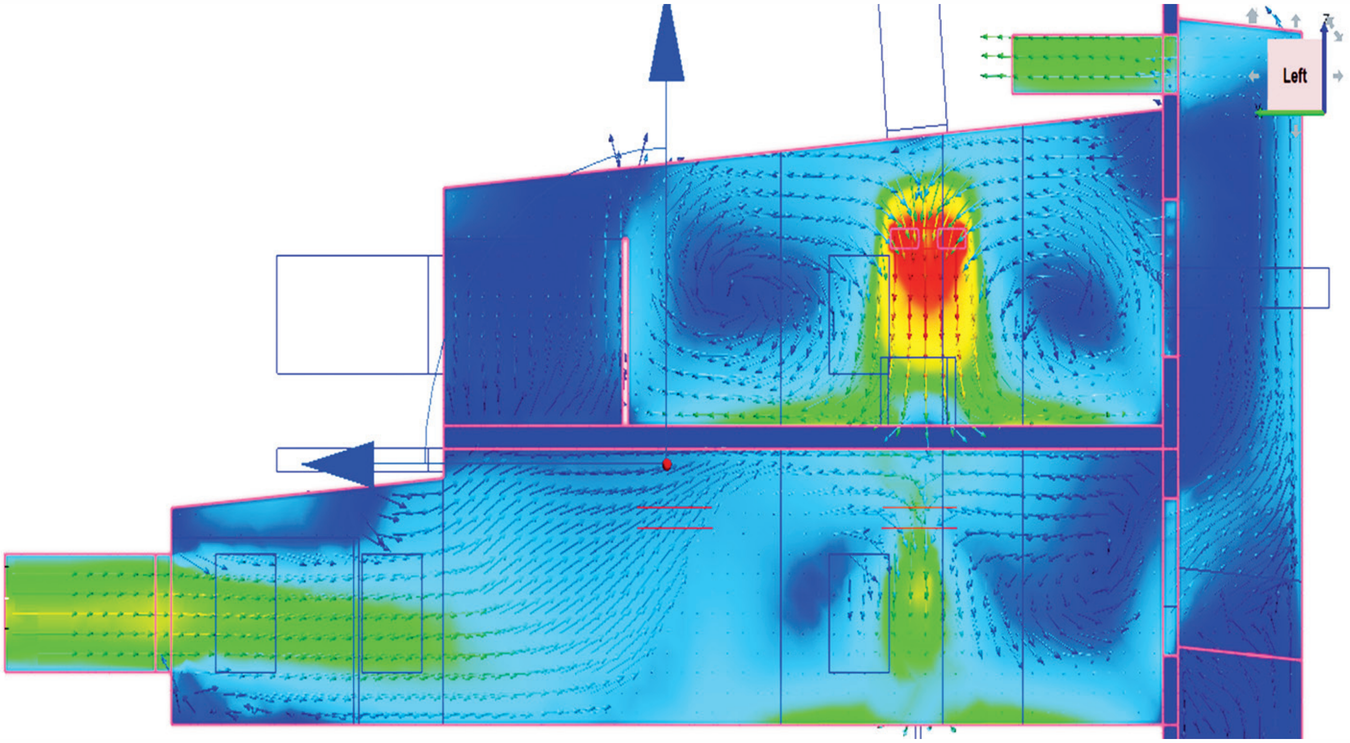
Air flow velocity simulation for a two-story multipurpose building via section-cuts



Using CFD in Building Design

By Dr. R S Maurya,
Professor, Mechanical Engineering
Sardar Patel College of Engineering, Mumbai

Abstract

With recent enhancement in computing capacity and rapid development in numerical algorithm, computational fluid dynamics (CFD) has emerged as a promising tool for design and analysis of a human comfort system. Globally, it is playing a significant role in planning and design of indoor and outdoor environment of major projects. All modern building designers, HVAC engineers, consultants and architects should be aware of the fundamentals of CFD processes, execution and interpretation of results. The present article discusses subject fundamentals and tries to highlight the essentials of a CFD analysis that need to be checked to ensure its quality. The article also tries to create awareness of the computational complexities, challenges and limitations.

Introduction

With increasing urbanization of population, the concentration of people in certain regions is changing rapidly. This has influenced the local environment in which the urban population spends most of their time. The limitation of land and natural resources has caused the development of high-density built environments to accommodate more people. Such environments have changed the availability of some key natural resources such as sunlight, wind flow, rains, rain water flow, etc. The shape of buildings, geometry, orientation, site coverage, façade area, height, layout and the permeability evolved due to specific arrangement, all affect the environment. Also, massive dense structures behave like local heat sources, which has the potential to change the micro-environment in terms of temperature distribution, wind flow pattern and air quality.

Disturbed outdoor natural environment affects indoor environments, which costs more in terms of energy use. This fact has been proved by an energy survey, which shows that the developed and developing world together consume more than 40% of the total energy flow to maintain a desirable indoor environment. As people spend more than 90% of their time in indoor environment, it is desirable that indoor and outdoor environment must be designed simultaneously to optimize energy consumption. Urban designers, city planners, site developers, architects and engineers can play key roles in the development of future cities and contribute significantly in the existing expanding cities.

About the Author

Dr. R S Maurya has a doctorate from IIT Madras, and is a faculty at SPCE. He has more than 20 years of experience in academics and research. Being deeply involved in using computational fluid dynamics (CFD) for HVAC applications, he has executed many industrial projects, as well as academic projects at UG, PG and doctorate level.

This can be achieved by proper investigation of both indoor and outdoor environments. The conventional design methodology depends on crude fundamentals, observations, thumb rules, codes and standards. It contributes to meet the objectives to a certain extent, but fails to meet the design requirements beyond a point. A computation based technique, Computational Fluid Dynamics analysis, has the capability to reveal many of the secrets which conventional techniques fail to do.

In recent years, CFD has proved itself as a potential and promising tool for planners, designers, engineers, consultants and architects for successful execution of their projects. It has been playing an increasingly important role in the design environment. *Blocken et al.* [1] have discussed the potential application of CFD to predict outdoor environment. In 2006, *Zhai* [2], reviewed the application of CFD for indoor and outdoor environment design and enumerated the development in this direction. Some common situations where CFD analysis can help us significantly are:

- Where acquiring experimental data is difficult, costly or time consuming.
- Where sensitivity test of influencing parameter is desired.
- Critical projects, when neither similar experience nor measured data exist (such as large spaces, unconventional ventilation system, strong buoyancy effects).

CFD provides spatial and temporal distribution of airflow, pressure, temperature, turbulence-intensity, moisture and contaminant concentration.

Some examples of outdoor environment problems are:

1. City planning, cluster development and buildings
2. Impact of building on its surrounding environment
3. Pollutant transport and dispersal
4. Design of pedestrian spaces

Some examples of indoor environment problems are:

1. Indoor environment quality – room air movement, air quality, concentration distribution, containment and pollutant transport, thermal comfort assessment, ventilation system effectiveness, etc.
2. HVAC system and equipment design – duct design, air curtains, diffusers, exhaust hoods, cooling tower, compressor, condenser, evaporators, etc.
3. Smoke and fire management in buildings.

The objective of the present article is to make the consultants, designers, engineers and architects aware of:

- The potential of CFD in indoor and outdoor environment building design
- The process of CFD analysis
- The complexities involved in the investigation
- The process of verifying, checking and judging the report of a CFD work

This understanding would be helpful to understand the requirement of a CFD analysis in a project, providing support of essential data for execution of work if work is assigned to others and checking the quality of CFD work done.

Process of CFD Analysis - Fundamentals and Execution

CFD as a research tool was born in the research laboratory to solve hardcore complex engineering problems. Slowly it expanded its wings to serve other fields of engineering. Its success led to the development of a number of commercial software. This attracted many people and now a pool of trained people capable of using such software is available. CFD is not limited to knowing the software to run it. It is much more than that. A CFD analyst must have good understanding of fundamentals. It is an amalgamation of many fundamentals such as thermodynamics, heat and mass transfer, computational techniques and subject areas. CFD analysis is a well thought of exercise that needs preplanning, modeling, execution, interpretation and presentation of the results. The key steps are shown in flow chart in *Figure 1*.

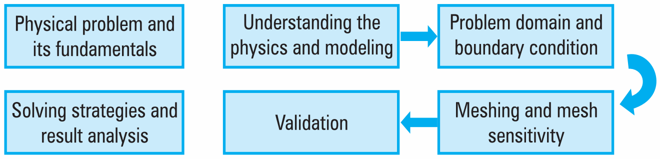


Figure 1: Process of CFD Analysis

Understanding the Physics and Modeling

Primarily, the success of a CFD investigation depends on a proper understanding of the associated physics. Most of the problems associated with indoor and outdoor environment are complex in nature due to the diversity involved in the scale of domain, range of velocity, temperature/density stratification, transient behavior, turbulence, evaporation or phase changes, presence of porous medium, multiphase, mixed mode of heat transfer, and random heat and/or mass generating sources. So, a CFD analyst should have good command on fundamentals, transport equations, different phenomenological models and different modeling techniques. It greatly helps in problem execution and correct interpretation of results.

Problem Domain and Boundary Condition

Domain selection to carry out the desired investigation is an intelligent decision that comes through observation and information available in the problem. It affects the result and computational time. Identifying inlet, outlet, heat source, heat sink and other significant boundaries are an essential part of a computational domain. Selecting an appropriate boundary condition is a very sensitive issue of problem execution. It significantly affects the result. A wrong boundary condition may either give misleading results or lead to divergence in the solution. Apart from using the regular boundary condition, all transient investigation needs the initial boundary condition also.

Meshing and Mesh Sensitivity

Spatial discretization is an inherent step of CFD analysis which is called meshing. The computational domain is converted into meshed domain by appropriately dividing it into a number of small cells. Type, quality, density and concentration of cells in a specific region is dictated by the physics of the problem and affects the result significantly. So, CFD works need to be checked for their independence from cell properties. Hence, mesh sensitivity test is essential to ensure reliability in the predicted result. A CFD investigated report must have the mesh sensitivity test also.

Validation

A CFD analysis is a solution of a model of a natural problem. Modeling is a hypothetical activity that results from multilayered assumptions and approximations made during the simulation process. All CFD work undergoes three distinct level of modeling – physical, mathematical and numerical. A reasonable model introduces lesser error in the result compared to others. Estimation of error induced in a practical problem due to modeling is a challenging task. To check the correctness and to quantify the error in a simulation model needs to be validated with available benchmark results. This is very essential to ensure reliability and to maintain confidence in the computed result. Generally experimental, analytical or a well established numerical result is considered as a benchmark for the validation. For the environment design, *Srebric and Chen* [3] suggested paying special attention to the basic flow and heat transfer, turbulence models, auxiliary heat transfer and flow models, numerical methods and assessment of CFD predictions during the validation process:

ASHRAE Handbook recommends that it is not necessary to start validation with a very complicated case if alternatives are available. Reliability is expected to be better

- for a simpler geometry, rather than a complicated one,
- for convection, rather than combined convection, conduction, and radiation,
- for single-phase flows, rather than multiphase flows,
- for chemically inert materials, rather than chemically reactive materials.

Figure 2 illustrates a validation, where experimental data and simulated result are compared. A good fit with little deviation in both results can be observed. It is a part of a CFD work done by *Virkute and Maurya* [4], where the objective was to assess the characteristic of flow in a variable spacing between two buildings.

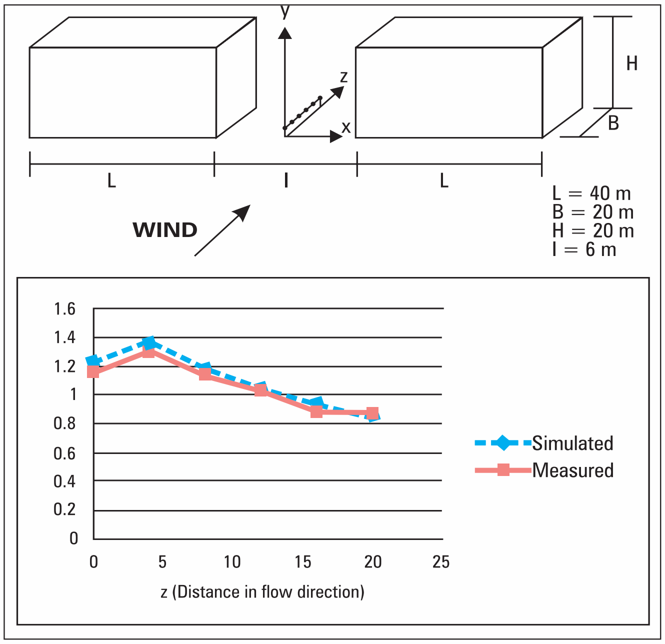


Figure 2: Illustration for validation of a CFD work

Solving Strategies and Result Analysis

While initiating the solution, the analyst has to take some crucial decisions about the nature of problem such as steady or transient, inviscid or viscous, laminar or turbulent, coupled or uncoupled, etc. Computation starts with iterative solution of discretized transport equation. In many CFD cases, several physical phenomena (different modes of heat transfer, turbulence, species transport, evaporation, combustion, etc.) simultaneously influence the system's performance. To solve such problems, the effect of these phenomena should be implemented in steps to ensure that they have been properly taken into account in the predicted solution. This approach simplifies complicated models and increases the probability of success. The result should be analyzed with the fundamentals. If the result is found to be contradicting the fundamentals, the solution process needs to be checked and repeated.

Potential Applications

CFD analysis can provide solutions to many thermal and flow problems of indoor and outdoor environment design, which is not possible through the conventional approaches. A few illustrations and discussions are presented in the following sections.

Design of Outdoor Environment

Increasing urbanization is slowly making our existing cities congested due to limited land resources. Denser and taller building structures are being designed to meet the requirement of rising population and land constraint. The outdoor environment is getting disturbed due to the following reasons:

- Low wind speed due to reduced permeability
- Change in air flow pattern near and surrounding the buildings
- Creation of urban heat island causing increase in urban air temperature *(Figure 3)*
- Reduced natural ventilation causing deterioration in air quality and human comfort
- Dispersal of pollution caused by vehicular transport and local industries
- Persistence of hot and humid environment throughout the year in coastal cities

Use of CFD can be useful in the assessment of these problems and providing solutions. It can optimize building sites by predicting the

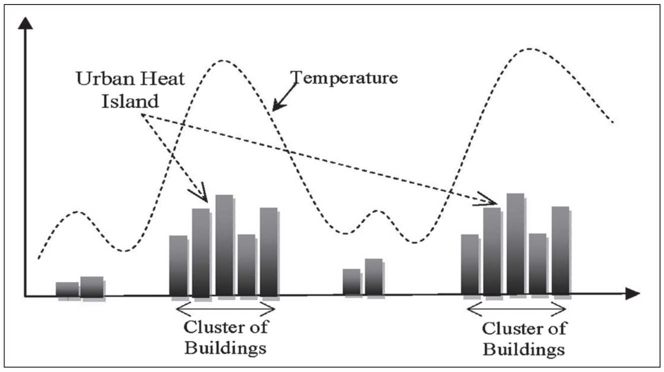


Figure 3: Development of urban heat island (UHI) in cities

continued on page 26

continued from page 24

Figure 4: Bird's eye view of the site

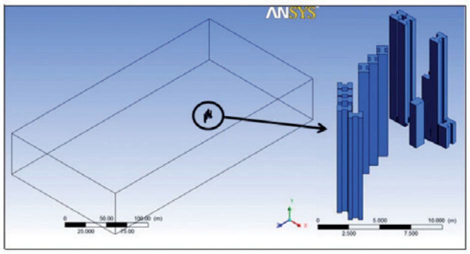
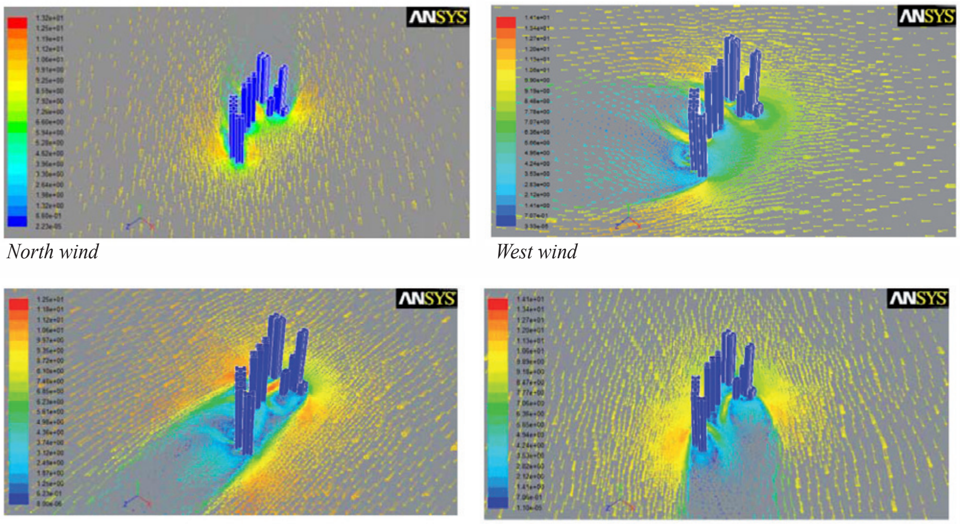


Figure 5: Computational domain and solid model



North wind

West wind

(c) South-west wind

(d) South wind

Figure 6: CFD for site planning and development (velocity vector)

wind flow pattern, temperature, moisture, pollutant concentration and turbulence in the flow around buildings. So, it can contribute significantly in city planning. The most challenging issue of this domain is involvement of multi-scale and multi-physics.

A CFD work of outdoor environment design done by *Virkute and Maurya* [4] is illustrated in *Figure 4* to *7*. The location is a site of the Mumbai region taken from the internet. The objective of the CFD work was to investigate the effect of wind direction on a cluster of buildings. The study was helpful in the assessment of air velocity distribution, identifying stagnant zone, estimation of pedestrian comfort level, etc.

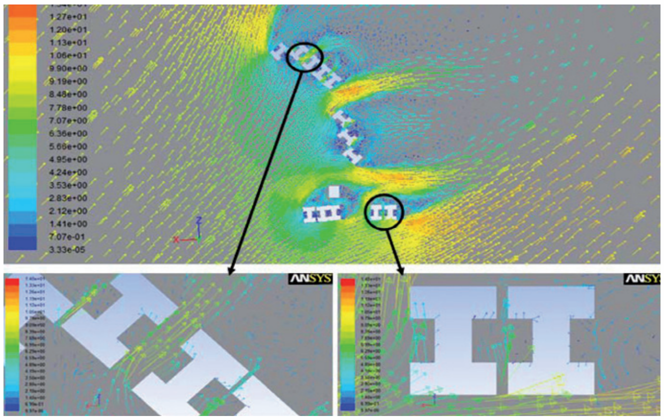


Figure 7: CFD for site planning and development (velocity vector)

CFD is capable of providing very detailed information such as local velocity, temperature and pressure. Such information is important in many applications, where control over micro environment is desirable. One such illustration is depicted in *Figure 7*. The pedestrian velocity can be predicted very well, which gives us the idea of optimum spacing in between the buildings. High pedestrian velocity creates uncomfortable environment for elderly people. Also, a stagnant zone near the building indicates existence of poor natural ventilation.

Design of Indoor Environment

Compared to the outdoor environment, CFD can contribute much more in the design of indoor environment. The major areas where it can contribute significantly are:

- Performance Analysis of a Ventilated Space or Building
- Pollutant Management
- HVAC System Design

Performance Analysis of a Ventilated Space or Building

Building sustainability is an issue of great concern in urban areas. CFD can play a significant role to achieve this objective. The effectiveness of new ideas can be assessed comfortably. Both indoor and outdoor environment can be simulated simultaneously to find an optimal ventilation strategy. Following are some of the common applications in indoor environment design, where CFD provides microscopic information of the key parameters.

i. Air diffusion and air movement analysis
ii. Thermal comfort assessment
iii. Indoor air quality prediction
iv. Prediction of contaminant dispersal
v. Effectiveness of a ventilation system
vi. Fire and smoke spread and control

Apart from the above applications, designers and architects can make use of CFD predicted information by assessing their visualization, ideas and concepts. Another significant domain is sensitivity study of key environmental parameters. Such investigations help in reducing the project cost.

An illustration of CFD analysis being used for indoor environment design is depicted in *Figure 8*. This work has been done by *Yadav and Maurya* [5] to investigate solar chimney assisted ventilation in an enclosure. The objective of the project was to investigate the effectiveness of different geometrical parameters on natural ventilation induced by a solar chimney. The computational domain, details of solar chimney streamlines of flow occurring in the enclosure

continued on page 28

continued from page 26

and the effect of solar heat flux on a ventilated mass is shown in *Figure 8 (a)* to *(e)*.

Pollutant Management

CFD can be effectively used to track the movement and propagation of pollutants, smoke or contaminants in or outside the space. As people spend most of their time in indoor activities, so it is more significant for indoor applications. The movement of contaminants depends heavily on several parameters such as geometry of space, structure of building, HVAC system used, partitions, furniture and passage-ways between indoor spaces. CFD helps in location optimization of control and sensing devices for their best performance. It is an intelligent tool for predictive studies in many more applications – chemical processing enclosures, engine laboratories, biological laboratories, operation theatres, isolation wards of hospitals, etc.

HVAC System Design

In order to achieve desirable indoor environment, several mechanical devices need to work in rhythm with their maximum system effectiveness. The conventional approach recommends use of thumb rules, charts, codes and standards with little experimentation. Such an approach succeeds most of the times, but fails to provide an optimum system performance due to associated uncertainties. CFD is an efficient tool to reveal many hidden facts which are crucial for effective and efficient system design and performance. An innovative design can be investigated for its effectiveness without converting thought into reality. Some applications are listed below.

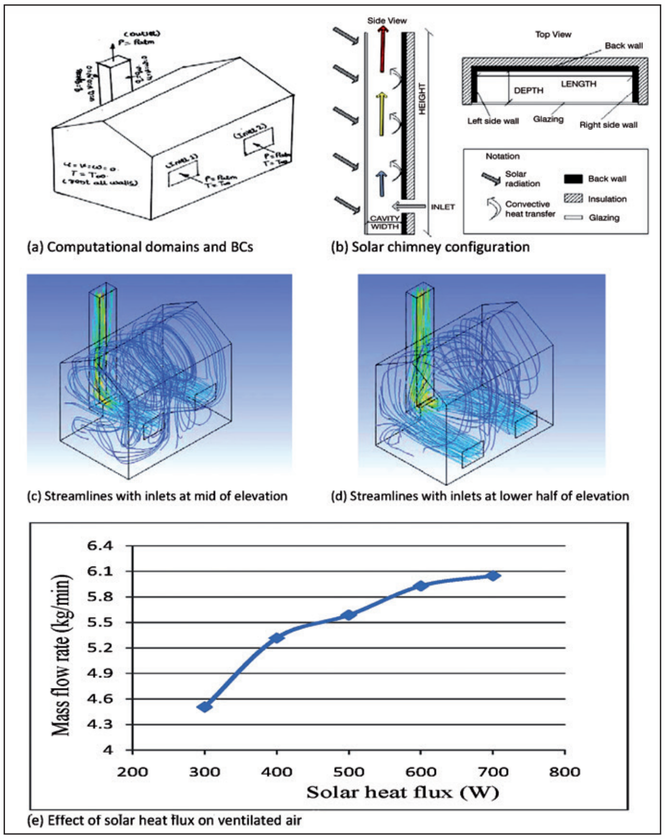


Figure 8: Solar chimney assisted natural ventilation

i. Investigation of ventilation arrangement, components and layouts,
ii. Diffuser and evaporators,
iii. Duct layout design including flow mal-distribution,
iv. Exhaust system,
v. Fan and blowers,
vi. Cooling towers,
vii. Effect of heat and containment sources and their location, etc.

Computational Challenges

CFD analysis of indoor and outdoor environment is challenging due to the scale and diversity of physics involved. Some of the points worth mentioning in this context are:

- Selection of appropriate computational domain
- Mesh management – resolution, distribution and quality
- Selecting and implementing appropriate turbulence model
- Judgment of realistic boundary condition
- Non-availability of established models for a few common physical phenomena
- Numerical experimentation
- Modeling errors

Conclusion

With the rapid development in computing power, CFD has now established its reputation as an efficient tool for predicting flow and heat transfer behaviour. Increasing number of research publications on indoor and outdoor environment design verify this fact. In the 2008 edition of ASHRAE Handbook (Fundamentals), an exclusive chapter on 'Indoor Environment Modeling' has been included to guide users. At different levels of projects execution, CFD can help building engineers, designers, consultants and architects to make their work more effective. But, care must be taken to verify the correctness of CFD predicted results before using them.

References

1. Bert Blocken, Ted Stathopoulos, Jan Carmeliet and Jan Hensen, 'Application of CFD in Building Performance Simulation for the Outdoor Environment'. Eleventh International IBPSA Conference, Glasgow, Scotland July 2009. 27-30.
2. Zhiqiang Zhai, 'Application of Computational Fluid Dynamics in Building Design: Aspects and Trends'. Indoor Built Environment. 2006. 15. 305–313.
3. J Srebric, and Q Chen, 'An example of verification, validation, and reporting of indoor environment CFD analyses'. ASHRAE Transactions. 2002.108(2). 185-194.
4. Tianzhen Hong, S K Chou, T.Y. Bong, 'Building simulation: an overview of developments and information sources'. Building and Environment. 2000. 35. 347-361.
5. Nilesh S. Varkute and R S Maurya, 'CFD Simulation in Township Planning– A Case Study'. International Journal of Computational Engineering Research. 2013. 3(3). 65-72.
6. Rohit K. Yadav and R S Maurya, 'Numerical Investigation of Solar Chimney Induced Natural Convection in Space'. Proceeding of International Conference on Environment and Energy, December 15-17, 2014. JNTUH. Hyderabad.