iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 9 Issue: III Month of publication: March 2021

DOI: https://doi.org/10.22214/ijraset.2021.33215

Study of Draft Angle in Kitchen Chimney Hood Cover for Optimum Performance

Zubairkhan Kittur[1], Dr. Krishna Kumar Dwivedi [2]
[1]*Ph.D. Research Scholar, Department of Mechanical Engineering, Oriental University, Indore, India*
[2]*Professor, Department of Mechanical Engineering, Oriental University, Indore, India*

Abstract: This study has been undertaken to investigate the effect of draft angle in kitchen chimney for the purpose of reducing energy wastage and optimum performance. The current study adopts the usage of Computational Fluid Dynamics (CFD) techniques for estimation and evaluation of flow physics around the kitchen chimney domain. The results yielded from CFD analysis shall be used for obtaining insights and providing suitable solutions to further improve the design of the kitchen chimney.
Keywords: Chimney, Kitchen, Hood, Ventilation, CFD.

I. INTRODUCTION

The outer cover of kitchen chimney hood performs the function of capturing the smoke and guiding it towards the exhaust vent. The draft angle of kitchen hood is a noteworthy parameter in the chimney even though its significance only remains when the generation of smoke is very rapid. Studies were conducted in order to estimate the optimum draft angle of kitchen chimney hood having the least pressure loss across the channel. CFD techniques are widely used for predicting the behaviour and design of ventilation system. This study focuses on the use of CFD to assist in optimization of kitchen chimney designs, with a hope to obtain possible solutions to the problem of indoor air pollution.

The current research work adopts simulation approach for data analysis. CFD is a robust numerical analysis tool which is widely adopted for simulating various processes related to HVAC industry. As a thumb rule, CFD cannot replace actual experiment but can surely reduce the number of experiments required for testing and prototyping phase. In addition to this, the overall cost and associated risks can be reduced marginally using CFD tools. It provides a virtual environment to visualize fluid flow attributes before creating it physically. CFD simulation of the smoke is performed for evaluating the effect of draft angle of kitchen chimney hood cover on the overall performance of the kitchen chimney.

The numerical models obtained from CFD are based on the basic equations of heat transfer and fluid dynamics. Simple approach along with assumptions are often used for obtaining a solution. The detailed flow information is not predicted, but instead a rough approximation of ventilation behaviour is obtained using these models. The empirical models and the analytical models are quite similar. These types of models are sometimes based on advanced computer simulations or experimental measurements instead of just basic heat transfer and fluid dynamics equation. Generally, any CFD process can be divided into three stages namely: Pre-Processing, Solution and Post-Processing.

II. GEOMETRY DETAILS

The geometry of the kitchen chimney hood cover varies based on the kitchen size, chimney type and chimney dimensions. The end adjacent to the cooktop is generally diverged in order to capture the maximum amount of smoke generated during cooking process. On the other hand, the opposite end is usually a straight section since it just performs the function of channelling the smoke or pollutants outside the kitchen environment.

The geometry of the kitchen chimney hood cover considered in current study is based on the dimensions of mid-size household kitchen chimney available in the market. The total vertical height of the hood is 1.2 meters. The top section is straight with a height of 1 meter.

The remaining 0.2 meters below is inclined based on the angle of each case study. Kitchen chimney hood cover with a draft angle ranging from 0 to 60 angle is considered in the current study. The schematic diagram shown in Figure 1 below shows the geometry details of kitchen chimney hood cover.

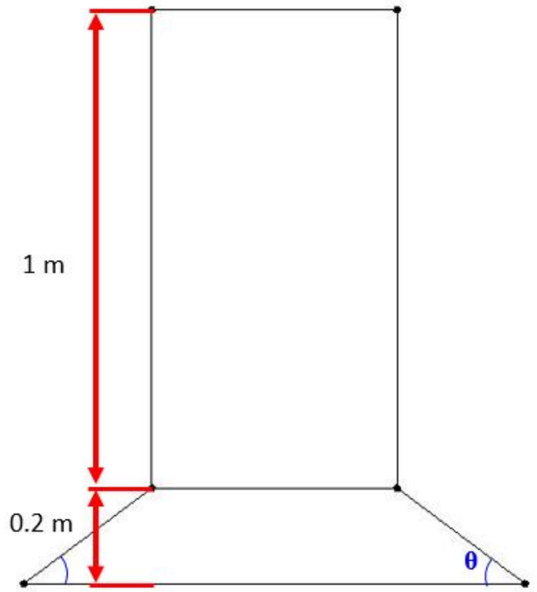


Fig. 1 Geometry details of kitchen chimney hood cover

III. BOUNDARY DEFINITION

The current CFD domain mainly comprises of inlet and outlet section. The smoke is propagated from the inlet section. It channels down to the vent or outlet section from where it is emitted to the external environment. The surrounding surface of the channel acts as a smooth wall. Figure 2 below shows the boundary modelling adopted in CFD analysis.

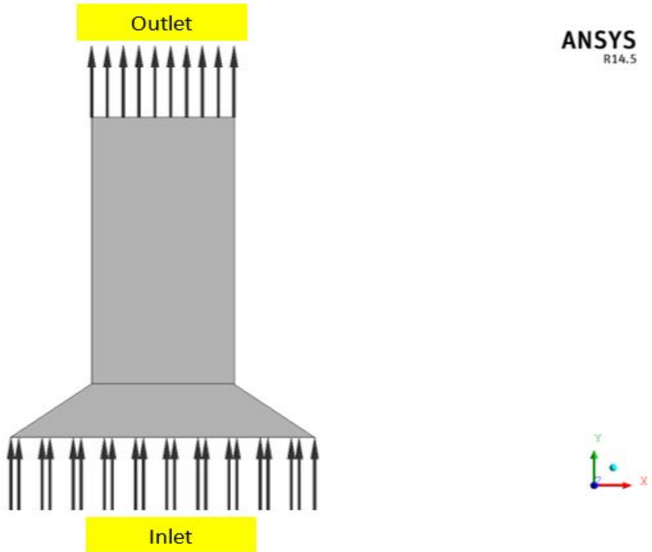


Fig. 2 Boundary modelling of kitchen chimney hood cover in CFD analysis

IV. MESH GENERATION

The CFD domain has been modelled and meshed using Ansys ICEM-CFD software. The modelled domain represents the kitchen chimney hood cover. Meshing has been done considering into account the boundary layer mixing phenomenon. The mesh near the wall is refined in order to capture the smallest flow data due to boundary layer mixing. Thus, the accuracy of the obtained results is increased and the flow physics can be captured more accurately. The region towards the core flow is defined with a coarser mesh since the flow is continuous in this region without any obstruction. The flow near the wall will slightly take more time to reach the outlet since it accounts for some resistance from the wall. The computation time is increased in such cases since fine mesh is defined near the wall regions. The coarser the CFD mesh, the less is its computation time.

On the other hand, fine mesh takes a lot more computation time and memory to perform CFD analysis. It should be noted that coarse mesh often generates inaccurate results and hence cannot be relied upon. Judgement should be taken based on the complexity of the model in order to achieve balance amongst computation time and mesh count such that there is no drastic trade-off on either side. Figure 3 below shows a sample of the mesh generated for the current research work.

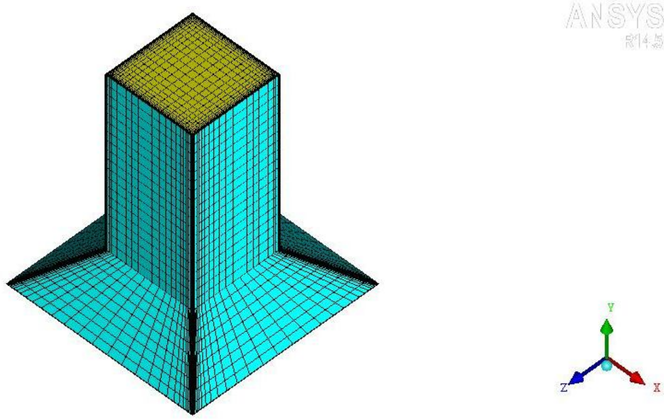


Fig. 3 Mesh generation on CFD domain of kitchen chimney hood cover

V. SOLVER SETTINGS

The meshed CFD domain has been solved by considering steady state analysis using Ansys-CFX software. The solution is based on the boundary conditions shown in Table I below.

Table I
Boundary conditions in chimney hood cover

Location	Boundary Condition
Inlet	Mass flow inlet (smoke-air mixture)
Outlet	Mass flow outlet (smoke-air mixture)
Other surfaces	No slip wall

The smoke present in the domain is defined as additional variable using a transport equation. Additional variables are commonly defined in terms of concentration. Such variables can either be transferred or added by another fluid. If such variables are modelled using transport equation, they get carried along with the fluid. It is assumed that the existence of smoke does not have any interactive effect on the air flow. Hence this definition is most suitable when the concentrations of smoke particles is small. The flow in such scenario is considered to be a multiple component fluid having two components namely smoke and air. The existence of smoke can however affect the flow of fluid based on the defined fluid properties. But however, the air and smoke are presumed to have similar velocity. Such models can widely be used for higher concentrations of tiny smoke particles. The flow in this case is considered as one single fluid (Air) along with smoke particles modelled using particle tracking.

The material for the hood cover is considered as Aluminium in this study. Hence standard material properties related to aluminium are used for analysing flow results. K-epsilon turbulence model has been considered since it provides good results for swirl flows. Turbulence based on default intensity and auto compute length scale are used since it works well with channel or conduit flow estimation. The effect of gravity is also considered for accurate prediction of the results. Using the above parameters, the domain is solved using a high-resolution scheme for 1000 iterations with double precision mode. The convergence criterion is set as 1e-06 for all the quantities. Generally, CFD results fall within 5% to 10% accuracy range of experimental results. Figure 4 below shows the CFD domain along with the defined boundary conditions in Ansys-CFX.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue III Mar 2021- Available at www.ijraset.com

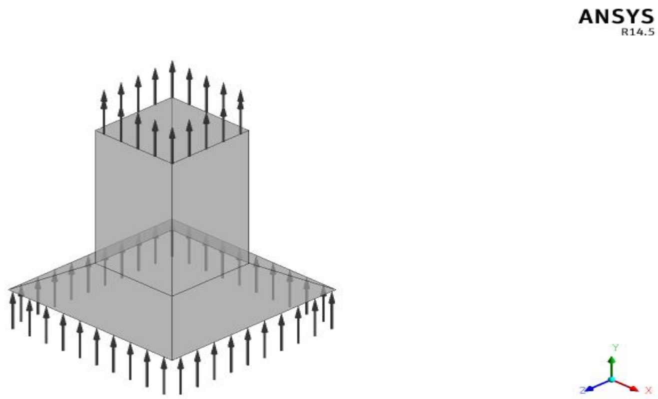


Fig. 4 CFD domain along with defined boundary conditions

VI. RESULTS AND DISCUSSION

Simulations were performed in order to assess the optimum draft angle of kitchen chimney hood having the least pressure loss across the channel. Draft angle ranging from 0 to 60 degrees angle have been evaluated in the current study. Generally, zero-degree (flat) draft angle hood covers are not recommended since the pressure loss is maximum in this case due to flow stagnation.

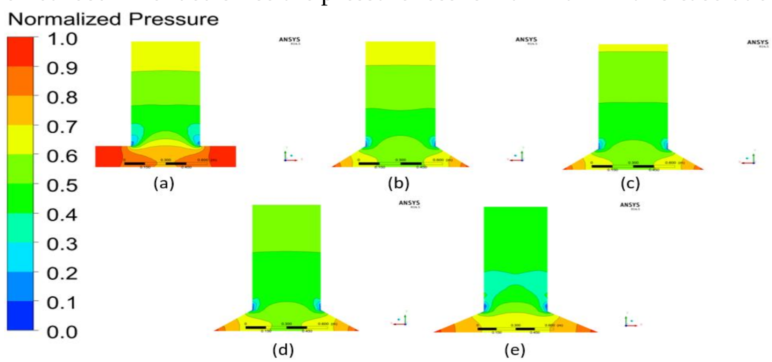


Fig. 5 Pressure contours for (a) flat or zero-degree configuration chimney (b) 35^{o} Draft angle configuration chimney (c) 40^{o} Draft angle configuration chimney (d) 45^{o} Draft angle configuration chimney (e) 60^{o} Draft angle configuration chimney

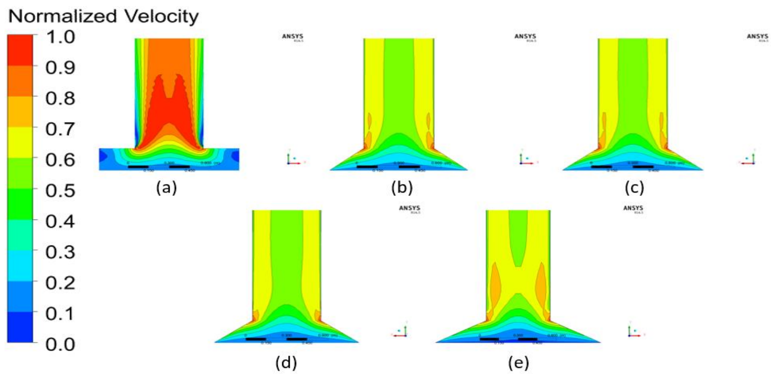


Fig. 6 Velocity contours for (a) flat or zero-degree configuration chimney (b) 35^{o} Draft angle configuration chimney (c) 40^{o} Draft angle configuration chimney (d) 45^{o} Draft angle configuration chimney (e) 60^{o} Draft angle configuration chimney

Figure 5 and Figure 6 represent the flow pressure and flow velocity contours for different draft angles of chimney hood cover ranging from 0 to 60 degrees. It is quite evident that kitchen hoods with flat hood configuration (zero-degree draft angle) are very inefficient in capturing the smoke. The reason for this is the pressure loss created due to high stagnation pressure across the sides. This results in generation of wakes which often leads to surge or reverse flows. Hence, the flow gets choked inside the channel and does not travel to the outlet easily. This problem can only be overcome by using high suction capacity motors and hence are not a viable solution in this research study.

VII. CONCLUSION

Qualitative analysis shows that kitchen chimney is efficient roughly between at 40 to 50 degrees draft angle of kitchen hood cover. The velocity contours of 45^o draft angle configuration chimney shows smooth flow of smoke-air mixture from inlet to outlet without and presence of wakes or eddies at the junction of the channel. Moreover, it also shows the least pressure loss across the channel. This shall also result in less flow resistance while travelling from inlet to outlet.

Figure 7 below shows the result of quantitative analysis. The pressure loss signifies the wake generation due to stagnation pressure. It is observed that the draft angle between 44 to 46 degrees is the most efficient configuration to drive smoke out of kitchen, having the least pressure loss of around 50.76% at the ends. Rest other draft angle configurations have higher pressure losses towards the edges which result in eddy or wake dissipation thereby degrading the performance of the kitchen chimney.

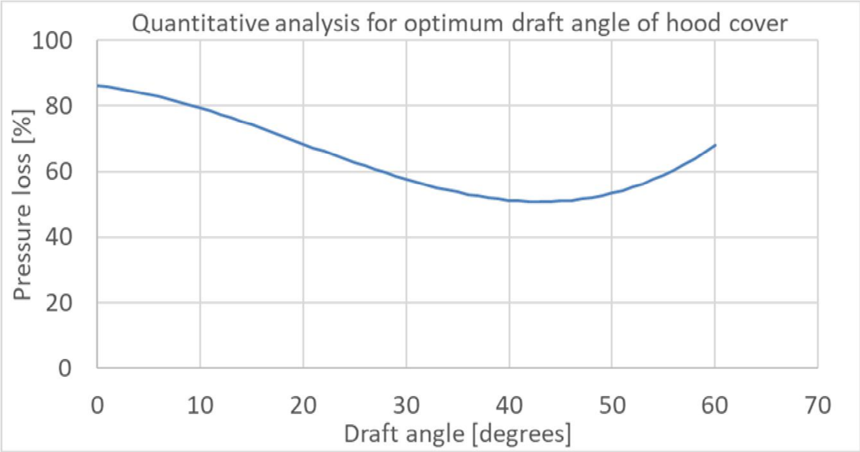


Fig. 7 Quantitative analysis for optimum draft angle of kitchen chimney hood cover

VIII. ACKNOWLEDGMENT

This research work was supported and guided by Dr. Krishna Kumar Dwivedi, Professor in Mechanical Engineering department of Oriental University, Indore, Madhya Pradesh, India. I would like to thank the entire management and faculty of Oriental University, Indore who provided insight and expertise that greatly assisted this research work. Without this support and guidance, this paper could not have reached its goal.

REFERENCES

[1] J. Chen, B. Duffy, D.F. Fletcher, B.S. Haynes and P. Nelson, FTIR spectroscopy measurements and CFD simulations of the pollutants arising from unfuelled combustion in a room, Building and Environment Volume 36, Issue 5, June 2001, Pages 597-603.

[2] Chi-ming Lai, Experiments on the ventilation efficiency of turbine ventilators used for building and factory ventilation, Energy and Buildings Volume 35, Issue 9, October 2003, Pages 927-932.

[3] Risto Kosonen, The effect of supply air systems on the efficiency of a ventilated ceiling, Building and Environment Volume 42, Issue 4, April 2007, Pages 1613-1623.

[4] Kyoungbin Lim and Changhee Lee, A numerical study on the characteristics of flow field, temperature and concentration distribution according to changing the shape of separation plate of kitchen hood system, Energy and Buildings Volume 40, Issue 2, 2008, Pages 175-184.

[5] Hwataik Han, Cheol-Yong Shin and Chang-In Baek, Correlations of control parameters determining pressure distributions in a vertical exhaust shaft, Building and Environment Volume 45, Issue 9, September 2010, Pages 1951-1958.

[6] Steven Nabinger and Andrew Persily, Impacts of air tightening retrofits on ventilation rates and energy consumption in a manufactured home, Energy and Buildings Volume 43, Issue 11, November 2011, Pages 3059-3067.

[7] Shubhajyoti Saha, Abhijit Guha and Subhransu Roy, Experimental and computational investigation of indoor air quality inside several community kitchens in a large campus, Building and Environment Volume 52, June 2012, Pages 177-190.

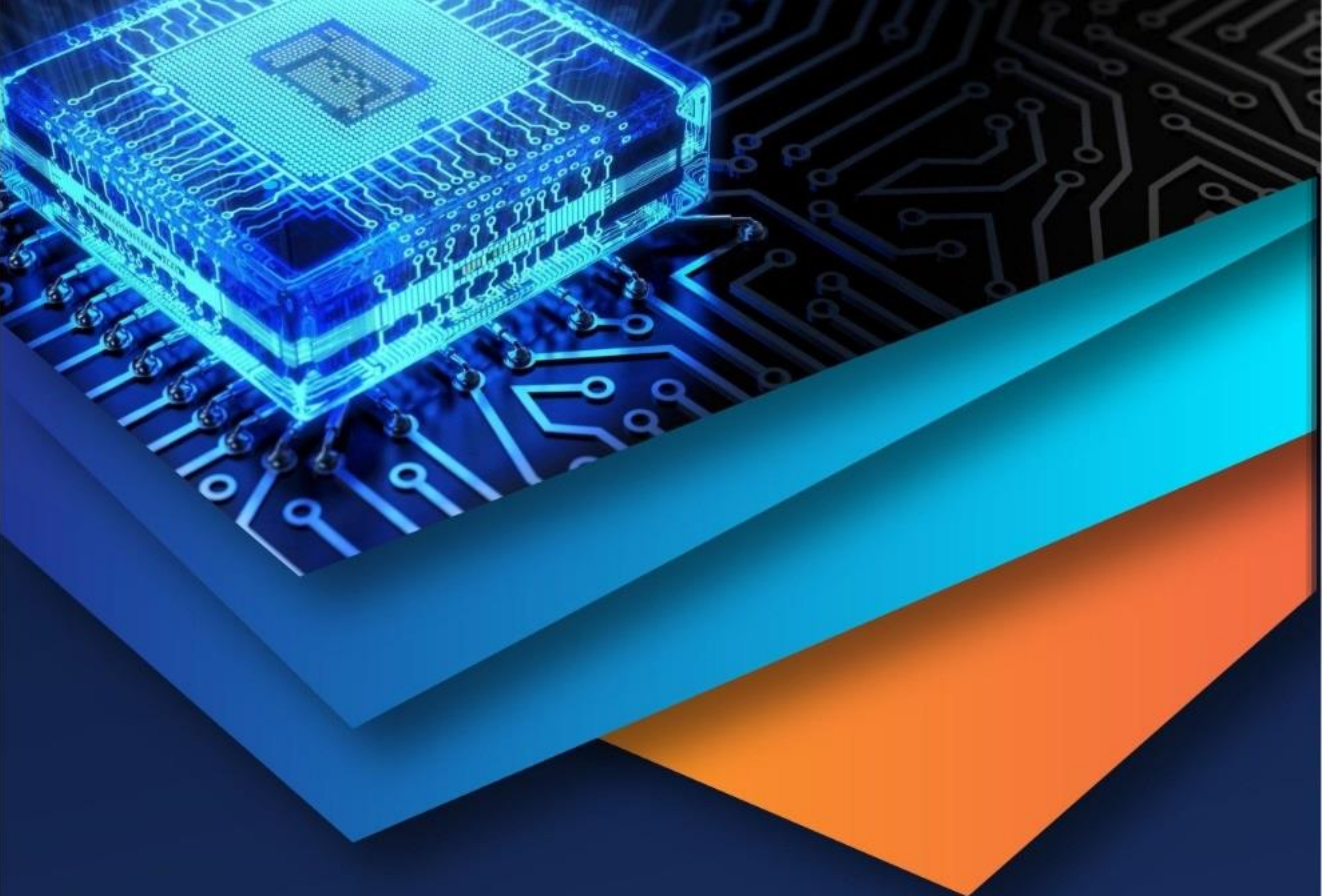
[8] Angui Li, Yujiao Zhao, Dahua Jiang and Xiaotan Hou, Measurement of temperature, relative humidity, concentration distribution and flow field in four typical Chinese commercial kitchens, Building and Environment Volume 56, October 2012, Pages 139-150.

[9] I.Dauta, N.Gomesha, M.Irwantoa,Y.M.Irwana, Y.Yanawati and S.NorShafiqin, Smart smoke ventilation and power generation (SSVPG), Energy Procedia Volume 18, 2012, Pages 1372-1381.

[10] Fong, K. F. and C. K. Lee, Towards Net Zero Energy Design for Low-Rise Residential Buildings in Subtropical Hong Kong, Applied Energy 93, 2012, Pages 686-694.

[11] Jun Gao, Changsheng Cao, Qianfen Xiao, Bin Xu, Xiang Zhou and Xu Zhang, Determination of dynamic intake fraction of cooking-generated particles in the kitchen, Building and Environment Volume 65, July 2013, Pages 146-153.

[12] Abdullahi, K. L., Delgado-Saborit, J. M., & Harrison, R. M., Emissions and indoor concentrations of particulate matter and its specific chemical components from cooking: A review, Atmospheric Environment 71, 2013, Pages 260-294.



10.22214/IJRASET

INDEX COPERNICUS 45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY