Comparative Investigations and CFD Analysis of Flue Gas Flow through Novel Kitchen Chimney Design

Mahesh G
Saranathan College of Engineering

Valavan D
Saranathan College of Engineering

Baskar N
Saranathan College of Engineering

Bovas Herbert Bejakhin A (✉️ herbert.mech2007@gmail.com)
Saveetha University

Research Article

Keywords: Novel kitchen chimney, Kitchen Ventilation, Chimney Advancements, CFD, Flue Gas flow

Posted Date: October 24th, 2022

DOI: https://doi.org/10.21203/rs.3.rs-2179779/v1

License: ☝️ This work is licensed under a Creative Commons Attribution 4.0 International License.
Read Full License
Abstract

Environmental concerns become increasingly evident as technology progresses and the world's population expands. Kitchen ventilation is a subdivision of ventilation that deals with the treatment of air from kitchens. Grease, fumes, and odours are all problems that kitchen ventilation solves. As a result, a cost-effective chimney was chosen to assist civilians. The primary components for venting the kitchen air are the filter, fan, flue, and duct. In this work, the new geometry duct design is improved and analysed by ANSYS. The volume of hot flue gas and discharge of hot flue gas are calculated theoretically and also recommended standard dimensions and geometry to the industry. Academic contributions to chimney advancements are recognised in this study endeavour, and a research deficit is identified based on these objectives.

Introduction

Chimneys are ventilation tunnels that direct smoke and other products of combustion out of a fireplace through the building's roof. It helps to remove oil, combustion products, fumes, odours, heat, and steam from the air when cooking by using air evacuation and filtration. In commercial kitchens, exhaust hoods are frequently used in conjunction with fire suppression systems to ensure that fumes from a grease fire are adequately evacuated and the fire is rapidly put out. The fans are fixed in the duct for sucking fresh air form the kitchen zone. High-performance ventilation systems significantly lower pollution levels while maintaining high interior air quality. This ventilation system protects the home's structure from hot gases passing through it, as well as the extreme temperatures that can arise from a chimney fire. The residential chimneys are divided into two types: those that serve fireplaces and those that serve appliances. While these two types vary in several ways, they both accomplish the same necessary functions.

Two major causes contribute to insufficient ventilation in a domestic kitchen. First, the parametric design of kitchen ventilation is poor, and ventilation technology is insufficient, unstructured, and unspecialized. The second reason for the inadequacy is that kitchen ventilation is not given enough consideration during the design and use phase. It is required to undertake a review of the offered technologies, evaluation methodologies, parameter ranges, and design approaches of the ventilation in residential kitchens in order to guide the better improvement of the kitchen interior environment. Khan Kittur and Piyush Jaiswal [1] considered proper ventilation of the Kitchen, a subsection of ventilation, is afraid with the outlet air from kitchen zone. Kitchen ventilation eliminates grease, smoke, and odours that aren't found in other types of ventilation. An extractor hood or canopy, as well as a filtration system, are included in kitchen ventilation equipment. The fan for the ventilation system can be placed in the kitchen or in a nearby duct system, depending on which is most convenient. This study acknowledges academic achievements in the field of kitchen hoods, and it proposes research gaps and new research targets based on these contributions.

Haiwei Jing and Angui Liv [2] research was done to forecast airflow rate, temperature field, and velocity field for various chimney gaps and heat fluxes. According to the findings, the width-to-height ratio of a
vertical solar chimney is excellent for reaching maximum airflow rate. The optimal ratio is 1:2. The temperature and velocity fields of the solar chimney channel were studied at this period. The air temperature and velocity are higher near the heated wall's surface than further away. A.T. Layeni and M.A. Waheed [3] the solution(s) to issues about electric power supply are a serious worry in many regions of both developing and growing countries. The Solar Chimney is proposed as a potential solution to this issue (SC). This research looks at the possibilities of using the Solar Chimney to ease Nigeria's energy issue while simultaneously providing ventilation in buildings. The Solar chimney (SC) is a natural flow device that adds heat to flowing air by converting the sun's solar thermal energy into kinetic energy in the air.

Jing Zhang et al [4], investigated the critical ventilation and controlling the food industries. The performance of the ventilation system is involved in this paper, it is reviewed and summarised the various methods ventilations. Long Shi et al [5] presents a world-first solar chimney design for a real structure that takes energy efficiency and fire safety into account. A mathematical tool was used to analyse characteristic design structures selected after validated by testing. A solar chimney may offer 7.42 air changes per hour natural ventilation and at least 6.52 times the available safe evacuation time for persons in common fire scenarios under normal conditions. The previously established ideal cavity gap of 0.2–0.3 m for a big space is no longer applicable, with the optimum gap for a large space being 1.2 m when both functions are included. Ramin Mehdipour et al [6] evaluates the thermal performance of solar chimneys experimentally under a number of experimental scenarios. When compared to previous experimental settings, the experimental sets employed in this work have two notable differences. This study illustrates the thermal and hydraulic performances of common types of sun chimneys, and low thermal performance of collectors is one of the key causes of solar chimneys' poor power production. When the collector shape is modified from circular to an innovative square shape, the Nusselt number (Nu), air flow velocity, and convective heat transfer coefficient (h) increase.

Han, H et al [7] investigated the uneven pressure distribution is calculated in vertical exhaust shaft. The performance and analysis of various parameters are discussed this this work. The new simulation software is developed and implemented various analysis. Saha, S et al [8] deals with concentration of CO₂ in the kitchen room size of 1.8m to 1.5m. The hot gases flow through multiphase flow is analysed by simulation software. Li, A., Zhang et al [9] studied about 32-storey residential building which employed central exhaust system and also investigated Static pressure and velocity distribution for the exhaust shaft with uniform cross section. Liu, Q et al [10] calculated the multiple burning sources in the specified kitchen dimensions by numerical methods. The computational Fluid dynamics calculations are used for this investigation. It is examined the various temperatures are calculated and analysed by numerical simulation. Ang xu et al [11] studied the contaminant attention for a kitchen cooking area. The various distribution of temperatures are analysed throughout kitchen cooking area. The comparative analyses were carried out the optimum oil fume pollutants in the kitchen area. Changshenge Cao et al [12] investigated the improvement of kitchen ventilation and reduced the air flow in the kitchen. There are two different modes are implemented and analysed the air flow in the kitchen area. Hyunsoo Lee et al [13]
investigated the ventilation behaviour of flue gas flow in the residential kitchen. The focussed 10 samples are selected and analysed the CO$_2$ value by simulation process. It is mainly focused on optimum ventilation for the Korean housing buildings. K.L. Abdullahi et al [14] analysed the concentration of particulate matters and emission level in the cooking kitchen. The various chemical matters are analysed and selected proper composition. Sumei Liua et al [15] calculated the new ventilation system in housing kitchen. The quality of air and environmental parameters like air temperature, CO, CO$_2$, TVOCs, and PM2.5 are analysed by simulation software. Che-Ming Chiang et al [16] analysed the modelling and monitoring natural ventilation in the taiwan kitchen. The various porosities are calculated by computational fluid dynamics method. Zhenlei Chen et al [17] investigated the air quality, thermal comfort analysis and environment situation. The CFD model delivers a practical tool to estimate the thermal comfort and air quality of the kitchen environment.

Chen, Wenhua et al [18] analysed the large scale ranges of PIV measurements of hood driven flow in the kitchen. It is compared with hood-driven flows under isothermal and heating conditions. The spill appearances of the cooking flue gas, instantaneous air distributions and vortex identification were analyzed. Ann Kristin Sjaastad and Kristin Svendsen [19] analysed comparison of four dissimilar types of canopy hoods in common use in private Norwegian households. The mass concentration particles are analysed in various dimensions of kitchen area. Kyoungbin Lim and Changhee Lee [20] discussed the characteristics of fluid flow and temperature analysis in the kitchen hood system. It is analysed three different models of kitchen hood system. The high efficient temperatures were analysed 1.4–1.9% and CO$_2$ concentration distribution is achieved at 9.4–11.9%. The CFD study of the newly created nozzle design achieved the superior results with flow rate decrease when compared to the existing water tap nozzles, according to A B H Bejaxhin et al.[21] The simulation aids in the prediction of each small hole's cross section and flow capacity.

The rectangular-shaped residential chimney that is listed here as the existing design has been the subject of numerous criterion reviews in all of the study articles. In this study, a novel kitchen chimney design has been created, and calculations have been made to compare the flue gas flow rate, effective time, velocity, and pressure for the rectangular and suggested geometries. It is also necessary to compare the temperature flow through the proposed and existing chimneys. The aforementioned parameters are used to compare the suggested model to the current one.

**Experimental Methods**

The rectangular-shaped chimney is the most prevalent. A comparison examination is performed based on the rectangular shape of the chimney. The current chimney's rectangular cross section is taken as 600mm X 500mm and the different views of exist kitchen chimneys are shown in Figure 1, 2 & 3.

The proposed chimney model is shown in the image below. This model has an ellipse cross-sectional area as an intake and a circular duct. The major and minor axes of the elliptical bottom measures 600mm and 500mm respectively and the different views are shown n Figure 4, 5 & 6.
The fundamental advantage of the suggested model is that it lacks the harsh edges found in the old model, resulting in a smoother and faster flow of flue gas. The height of the duct in the proposed model is also reduced (a part of design modification) which enhances the velocity of the flue flowing through the chimney.

*Analytical calculation for Rate of flow of flue gas*

**a) Existing chimney**

Discharge = Area(A) * Velocity(V)

⇔ \( Q = A \times V \)------------------------ 1

⇔ \( A_{1, \text{existing}} \times V_{1, \text{existing}} = A_{2, \text{existing}} \times V_{2, \text{existing}} \)------------------------ 2

where \( A_{1, \text{existing}} = \text{Inlet area of existing chimney (m)} \)

\( A_{2, \text{existing}} = \text{Outlet area of existing chimney (m)} \)

\( V_{1, \text{existing}} = \text{Inlet velocity of existing chimney (m/s)} \)

\( V_{2, \text{existing}} = \text{Outlet velocity of existing chimney (m/s)} \)

\[(0.5 \times 0.6) \times 1.2 = (0.18 \times 0.15) \times V_{2, \text{existing}}\]

\( V_{2, \text{existing}} = 13.33 \text{ m/s} \)

**b) Proposed chimney**

Discharge = Area(A) * Velocity(V)

⇔ \( Q = A \times V \)

⇔ \( A_{1, \text{proposed}} \times V_{1, \text{proposed}} = A_{2, \text{proposed}} \times V_{2, \text{proposed}} \)

where \( A_{1, \text{proposed}} = \text{Inlet area of proposed chimney (m)} \)

\( A_{2, \text{proposed}} = \text{Outlet area of proposed chimney (m)} \)

\( V_{1, \text{proposed}} = \text{Inlet velocity of proposed chimney (m/s)} \)

\( V_{2, \text{proposed}} = \text{Outlet velocity of proposed chimney (m/s)} \)

\[(\pi \times 0.35 \times 0.3) \times 1.2 = (\pi \times 0.075^2) \times V_{2}\]

\( V_{2, \text{proposed}} = 22.4 \text{ m/s} \)
Maximum volume of flue gas

a) Existing chimney

• Volume of trapezoidal hood:

\[ V_1 = \frac{1}{2} (b_1 + b_2) \times h \]  
\[ = \frac{1}{2} (0.6 \times 0.18) \times 0.5 \times 0.17 \]
\[ = 0.00459 \text{ m}^3 \]

• Volume of cuboidal duct:

\[ V_2 = l \times b \times h \]
\[ = 0.53 \times 0.18 \times 0.15 \]
\[ = 0.01431 \text{ m}^3 \]

• Total volume:

\[ V_{existing} = V_1 + V_2 \]
\[ = 0.00459 + 0.0143 \]
\[ = 0.01889 \text{ m}^3 \]

b) Proposed chimney

Volume of Truncated elliptical cone:

\[ V_1 = \frac{\pi b_h}{3a_1} (a_1^2 + a_1a_2 + a_2^2) \]
\[ = \frac{\pi \times 0.3 \times 0.5}{3 \times 0.35} (0.35^2 + (0.35 \times 0.075 + 0.075^2)) \]
\[ = 0.06920 \text{ m}^3 \]

• Volume of cylindrical duct:

\[ V_2 = \pi r^2 h \]
\[ = \pi \times (0.075^2) \times 0.2 \]
\[ = 0.00353 \text{ m}^3 \]

• Total volume:

\[ V_{proposed} = V_1 + V_2 \]
\[ = 0.06920 + 0.00353 \]
\[ = 0.07273 \text{ m}^3 \]
\[ = 72.73 \times 10^{-3} \text{ m}^3 \]

The below graph shows the comparison of volume of chimney occupied by the flue gas. It indicates that the volume of the proposed chimney is higher than the existing one as shown in Figure 7.
The maximum volume of proposed chimney is $72.73 \times 10^{-3}$ m$^3$ whereas for the existing chimney has $18.89 \times 10^{-3}$ m$^3$. The design modification and reduction in the duct's height in the proposed model leads to availability of more volume than the existing model.

**ANSYS Analysis**

Using ANSYS software, it is compared two chimneys that are now in use and those that are planned. It is extracting the required metrics, such as temperature, pressure, and velocity, from both the current and projected chimneys using ANSYS. The comparison of these parameters in our hands, and the graphs are retrieved to compare them graphically.

**Simulation methodology**

The simulation procedure starts with the construction of the kitchen chimney geometry and mesh using the Autodesk inventor and ANSYS software, respectively. One of the most crucial elements in generating an accurate simulation is meshing. A mesh is made up of elements that include nodes (space coordinates that change depending on the element type) that reflect the shape of the geometry. Meshing is the process of reducing irregular shapes into more recognizable volumes known as "elements". Mesh geometry has a considerable impact on the modelling of unsteady flow and turbulences. Mesh grows quickly from the initial mesh size. The local mesh size determines the minimal scale of unstable motion and turbulences accomplished in the simulation. The mesh specification should be sufficient to complete the flow feature, but the number of cells should be kept as low as feasible to retain the calculation's realistic speed.

As a result, the mesh density increases to the section of the mesh that is farther away from the fundamental mesh zone that requires a decent resolution. The meshing has been done for both the existing and proposed model. The Figure 7(a) and 7(b) shows the meshing of existing and proposed chimney using Ansys software. The meshing process here is automatic mesh breaks the model into several pieces to properly define the physical shape of the model for getting accurate result by taking the small portion of the model. The produced mesh must be re-examined to confirm that the meshing procedure was completed correctly. The border layer feature verification technique begins with the smallest elemental thickness and progresses to bigger elements with typical growth rates Mesh checks are conducted to ensure that the mesh is well established so that geometry definition may be completed.

After the geometry and meshing processes were completed, simulation and analysis using the CFD software proceeded. The chimney’s boundary conditions are determined using a number of assumptions that describe the broader system used in the simulation. The steady flow field motion phenomena are thought to occur on a short time period. With enhanced flow conditions and novel reactions, the walls of the combustion chamber are expected to attain quick thermal equilibrium. Without conduction of heat at the domain border, heat departs the domain over the wall surface along the interior of the kitchen chimney.
Results And Discussion

The simulation result can be displayed using the post processing step after all processing stages have been completed. The CFD simulation results may be allocated to any point in the chimney since it has a three-dimensional control volume. Some real findings, such as fluid flow and temperature patterns are analyzed and shown in 3D space as shown in Figure 8 (a) and 8 (b).

**CFD Simulation of Temperature levels**

It is necessary to concentrate three parameters to compare the temperatures of these two chimneys: inlet temperature, outlet temperature, and wall temperature. The inlet and the assumed to be as 323 K and 308 K respectively.

The wall temperature seems to constant throughout the chimney for both the cases as it is assumed to be as conduction free. Figures 9(a) and 9(b) depict the temperature pattern for the existing and suggested models. The orange-red zone in the image represents a temperature range of around 320 K to 323 K. In all situations, the maximum and minimum temperatures reached are 323.2 K and 308.1 K, respectively. The highest temperature is reached at the chimneys' entrance and exit. The bluish zone at the junction of elliptical hood and the cylindrical duct in the proposed chimney shows the temperature range of 311.2 K to 314.2 K.

**Velocity flow dynamics of chimneys**

The main metric for assessing the chimney is the velocity. The velocity is crucial in determining the chimney's efficiency. The entrance velocity is used as an input parameter to calculate the chimney's output velocity and is assumed as 1.2 m/s. The maximum velocity attained in the existing and proposed models of chimneys are 13.85 m/s and 20.36 m/s respectively. The velocity of the flue gas along the wall remains to be constant for both the models.

The yellowish-red zone in the Figure 10 (a) indicates the outlet velocity of the existing model ranges from 9.694 m/s to 13.85 m/s. In the case of proposed model, the outlet velocity ranges from 14.26 to 18.33 m/s which depicted in the yellowish-green zone in the following Figure 10(b).

As the cross-sectional area of the flue gas flow from the input to the output decreases, the velocity increases. In comparison to the existing chimney, the proposed chimney appears to have a smooth flow. The bottom half of the existing chimney, which has four edges, is replaced with an elliptical trammel form with no edges in the proposed chimney, resulting in a smoother flow. The suction component of the proposed chimney has a larger cross-sectional area, so more air flows through the intake. The suggested design allows the flue to flow swirlly around the chimney walls, resulting in a higher velocity in the model.

**Prediction of Pressure distribution on chimneys**
The principle of chimneys is that hot air rises above cold air. The movement of hot gases rising causes a pressure difference between the flue and the room. So, the pressure in the duct is one of the main parameters of kitchen chimney. The Figures 11(a) and (b) shows the pressure pattern in the chimneys as the shape is modified. The existing and suggested models have maximum pressures of 155.8 Pa and 80.62 Pa, respectively. The maximum pressure at the borders of the existing model ranges from 138.7 Pa to 155.8 Pa. The bluish-yellow zone at the intersection of the rectangular hood and the duct suggests a pressure range of 22.59 Pa to 79.67 Pa, whereas the recommended model predicts a pressure range of 22.03 Pa to 51.32 Pa.

The suggested model's elliptical hood encounters maximum pressure in the range of 70.85 Pa to 80.62 Pa. The pressure progressively drops to 25 Pa when the flue gas escapes the hood and reaches the duct entrance. The pressure in the duct does not fluctuate appreciably in either of the chimneys. According to Table 1, the existing chimney's observed flue gas flow rate of 13.33 m\(^3\)/sec is lower than the proposed chimney's predicted flow rate of 22.4 m\(^3\)/sec. It is obvious that the proposed chimney's rate of flow is larger than the chimney's existing design by a difference of 9.07 m\(^3\)/sec. For these types of home uses, a higher flow rate from this chimney outflow is better noted. Similar to this, the proposed model chimney design allowed for a maximum fluctuation in flow velocity of 6.51 m/s. The highest velocity of the proposed model can be seen to be 22.4 m/s, while the velocity of the present model was measured to be 13.85 m/s.

Table 1. CFD Analysis of existing and the proposed chimney design with variance

<table>
<thead>
<tr>
<th>Sl.No</th>
<th>Parameters</th>
<th>Exist Chimney Model</th>
<th>Proposed Chimney Model</th>
<th>Variance</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Rate of flow of flue gas (m(^3)/sec)</td>
<td>13.33</td>
<td>22.4</td>
<td>9.07</td>
</tr>
<tr>
<td>2</td>
<td>Velocity (m/sec)</td>
<td>13.85</td>
<td>20.36</td>
<td>6.51</td>
</tr>
<tr>
<td>3</td>
<td>Pressure (Pa)</td>
<td>70.85</td>
<td>80.62</td>
<td>9.77</td>
</tr>
<tr>
<td>4</td>
<td>Temperature (K)</td>
<td>323.2</td>
<td>311.2</td>
<td>12</td>
</tr>
</tbody>
</table>

The maximum pressure was also reached at the proposed new chimney design's intake and outflow, where it was 80.62 Pa, a difference of 9.77 Pa from the previous one. For this inquiry, the temperature ranges for the existing model and the suggested model are 323.2 K and 311.2 K, respectively. The temperature distribution has been reduced from the previous temperature distribution of 323.2 K to 311.2 K due to the elliptical shape of the chimney's cross section. This might be regarded as the difference between the rectangular chimney model and the model temperature, which is 12 K lower. Low temperatures increase the likelihood that a material will last a long time. Figure 12 depicts that the similar curves have been observed the values of CFD output parameters which is presented in the horizontal axis line. Comparatively, more differences have been indicated in between the proposed and existing model curve during the observations of flow rate, pressure and temperature. The increase of temperature
variation helps to identify the suitable model which can withstand the heat flow along the inner side of the chimneys and it was observed fine in proposed model chimney design.

**Conclusion**

In this project, it is presented a full assessment, including analyses of theoretical and numerical studies aimed at optimising the system's major characteristics, comparing and analysing the metrics such as velocity, temperature and pressure with respect to the existing system. The results show that the proposed model is more efficient than the existing model. The circular hood in the recommended model experiences maximum pressure, which gradually decreases to 25 Pa as the flue gas departs. It is clear that the proposed chimney's flow rate is 9.07 m$^3$/sec higher than the chimney's current design. A larger flow rate from this chimney output is preferably observed for these kinds of domestic purposes. In addition to that the elliptical cross section of the chimney has minimized the temperature distribution. The ability to find a model that can survive the heat flow along the inner side of chimneys is made possible by an increase in temperature variance, which was successfully seen in the proposed model chimney design.

**Declarations**

**Ethics approval and consent to participate**

- NA-

**Consent for publication**

On behalf of all co-authors, hereby I understand and declare that we have authorized and participated in this journal publication regarding of identifiable details, design data, figures and tables

**Availability of data and materials**

The data used to support the findings of this study are included in the article. Should further data or information be required, these are available from the corresponding author upon request.

**Competing interests**

The authors declare that there are no conflicts of interest regarding the publication of this paper.

**Funding**

Authors declared that no funding was received for this Research and Publication.

**Authors' contributions**

Dr. G.Mahesh – Research methodology and Novelty of the project
Acknowledgments

The authors appreciate the technical assistance to complete this experimental work from Department of Mechanical Engineering, Saranathan College of Engineering and Saveetha School of Engineering, SIMATS, Chennai, Tamil Nadu, India. The authors thank for the technical assistance to complete this experimental work.

References


2. Experimental Study of A Vertical Channel Solar Chimney With Uniform Heat Flux For Natural Ventilation In Buildings-Haiwei Jing, Angui Li.


Figure 1

Front View of existing kitchen chimney

Figure 2

Top view of existing kitchen chimney
Figure 3

Isometric view of existing kitchen chimney
Figure 4
Top view of proposed kitchen chimney

Figure 5
Front view of proposed kitchen chimney
Figure 6

Isometric view of existing kitchen chimney
Figure 7

Comparison of volume of existing and proposed chimney

Figure 8
Figure 7(a). Meshed model of existing chimney (b). Meshed model of proposed chimney

Figure 9

Figure 8 (a). Flow pattern of flue gas through existing model (b) Flow pattern of flue gas through proposed model

Figure 10

Figure 9 (a) Temperature pattern of existing chimney (b) Temperature pattern of proposed Chimney
Figure 11

Figure.10 (a). Velocity pattern of existing chimney (b). Velocity pattern of proposed Chimney

Figure 12

Figure.11 (a) Pressure pattern of existing chimney (b) Pressure pattern of Proposed chimney
Figure 13

Figure 12. Comparison of CFD output parameters of existing and new chimney.