Design of economic PODS to safeguard against contagious diseases using computational fluid dynamics (CFD)

Ankit Prakash¹, Nishant Sati¹, Piyush Pratap Singh¹, Sajal K. B. Degala¹, Anubhav Rawat¹, Ramesh Pandey¹, R.P. Tiwari¹

¹Applied Mechanics Department, Motilal Nehru National Institute of Technology-Allahabad, Prayagraj-211004, U.P., India

Abstract The outbreak of SARS-CoV-2, namely Covid-19, one of the greatest pandemics in human history posed some new challenges to the human beings for its prevention and control as it is an airborne respiratory disease and highly contagious. The most common practices suggested by WHO and government of India are: to wear a face mask, keep social distancing etc. It becomes much more important when the whole world is opening its economy. In such situations, there are a number of places where maintaining social distances or wearing a face mask is quite a tough job for example Laboratories and eating messes and canteens of schools and colleges. Thus, the present studies focus on the designing of economic PODS which may be helpful in controlling the spread of airborne disease in mass gathering places like in educational institutions in particular laboratories and eating messes. The present studies first emphasize on the flow field analysis of human cough by assuming it as a turbulent jet using CFD. Then, based on the physical dimensions covered by the coughing jets a further detailed study is conducted on 23 number of PODS of various shapes and sizes using CFD to select a suitable POD so that the effect of coughing in a laboratory/mess can be minimized to contain the infections. A POD with 1 m length and 30 cm height having an arc inside (POD 20) is found to be the best POD to contain the airborne infections.

1. Introduction

In December 2019, the outbreak of COVID-19, a major epidemic in human history, was caused by a novel corona virus that can be transmitted from person to person [1], which could lead towards community transfer in its vital stage. This needs to be controlled for the safety of society and humanity. At the time of writing, there have been 58 million confirmed infections and 1.4 million deaths worldwide attributed to COVID-19 [2] and the numbers are still increasing day by day. Current investigation has been taken up to design containment PODs to safeguard against infectious contagious diseases. To design these PODS, it is very important to understand the biology of virus and then simulation techniques of coughing flows. Severe acute respiratory syndrome (SARS) is a kind of epidemic disease which infects the human bodies by the tiny “droplets” or “particles”, termed uniformly as “particles” is a general rule of aerosol science [3]. These viruses or bacteria carrying “particles” are comprised of water, air, and pathogens generated by the respiratory activities of infected subjects [4]. Some of the infectious diseases of these categories are flu, measles, and mumps, etc. Respiratory activities like
coughing, sneezing, breathing, and talking are one the causes of the generation and dispersion of pathogen bearing droplets and aerosols [5]. Biological Sampling of these respiratory activities and studying it has confirmed the presence of viable pathogens within these “particles” [6–8]. The effect of pathogens are different on human beings, based upon the generation ways and duration. These pathogens containing particles generates in the form of droplets which begins to evaporate and decreases its diameter to few microns resulting in the suspension of these droplets in the air over a prolonged period [9]. Further the coughing jets may be considered as flow field similar to turbulent jets [10].

There are many important studies existing in the literature to simulate the human cough flows. Badeau et. al [11] predicted the cough particle characteristics using discreet element modelling of CFD with a focus on effect of human coughing on other humans in a square chamber. Gupta et al [12] studied the flow dynamics of cough on human beings by simulating the human mouth as cough emanating jets and evaluated the infectious source strength by coughing. Aliabadi [13] developed a Computational Fluid Dynamics simulation of near-field cough and sneeze droplet dispersion and considered various sources of variability in cough and sneeze processes. Thatiparti et al [14] studied a computational fluid dynamics study on the influence of the ventilation configuration on the possible flow path of bioaerosol dispersal behavior in a mock airborne infection isolation room. Bi [10] studied, LES and URANS approach of the Computational Fluid Dynamics (CFD) modelling of human cough flow and characterized in two different aspects, the flow field and the droplets to simulate the cough flow field. Recently Zohdi [15] developed a mathematical model to simulate cough particles analytically and numerically. This model comprises of range, distribution and settling time related to the particles under the influence of gravity and drag associated with the air surrounding it. Figure 1 and 2 clearly shows the flow field of coughing and a steady turbulent round free jet [16, 17].

![Figure 1 Schematic Flow field of coughing](image1)

![Figure 2 Schematic structures of a steady turbulent round free jet](image2)

The above literature survey reveals that the coughing is a phenomenon similar to the turbulent jets and thus before designing any containment measures to curb the spread of infectious diseases it is important to understand the flow characteristics of the coughing. Therefore, first the coughing jets have been modelled and simulated using CFD. Further, based on the zone affected a suitable design of PODS is predicted. Considering the economics and ergonomics the analysis of different shapes and sizes of the PODS have also been done so that the infected air does not escape the PODS.
2. Computational Strategy

Firstly, to reduce the computational complexity the flow is considered as 2-D, steady, continuous, incompressible, and single-phase, with nil gravity effects. The above assumptions have been made as they do not suppress the quality of designed PODs and saves much computational time and cost.

Since coughing flow field behaves as steady round turbulent jet, selection of an appropriate turbulent model is required. The most common turbulent models found to be used in literatures [18-25] are $k$-$\varepsilon$ and $k$-$\omega$. Here $k$-$\varepsilon$ model is preferred due to its applicability in free shear regions whereas $k$-$\omega$ model is suitable for near wall zones.

Also, $k$-$\varepsilon$ RNG model is preferred over standard $k$-$\varepsilon$ model as $k$-$\varepsilon$ RNG gives better results for both high and low Reynolds number in the same flow. Based on these attributes various researchers have also considered Renormalization Group (RNG) $k$-$\varepsilon$ over standard $k$-$\varepsilon$ for problems involving ventilation, airborne infection risk in healthcare studies [18-25].

2.1. Turbulence model and governing equations

Out of the various known turbulent models RNG $k$-$\varepsilon$ models is selected as mentioned and preferred in various literatures [18-25] for similar studies. The governing equations of the selected model are given below: momentum and continuity equations

\[
\frac{\partial \varepsilon}{\partial t} + (U_j) \frac{\partial \varepsilon}{\partial x_j} = C_{\varepsilon 1} \nu_T \left[ \frac{\partial (U_i)}{\partial x_j} + \frac{\partial (U_j)}{\partial x_i} \right] \frac{\partial (U_i)}{\partial x_j} - C_{\varepsilon 2} \frac{\varepsilon^2}{k} \left( \frac{\partial \varepsilon}{\partial x_j} \right) - S_{\varepsilon} \quad (1)
\]

where,

\[
S_{\varepsilon} = C_{\mu} \frac{\eta^3 (1-\eta/\eta_0) \varepsilon^2}{(1+\beta \eta^3) \varepsilon + k}\quad \text{and} \quad \eta = \frac{k}{\varepsilon} \sqrt{2S_{ij} S_{ij}}
\]

\[
\frac{\partial U_j}{\partial x_j} = 0 \quad (2)
\]

and $S_{ij}$ is the strain-rate tensor. The constants $\eta_0$ and $\beta$ take the values 4.38 and 0.012, respectively. This additional term is an ad-hoc model that is largely responsible for the differences in performance compared with the standard model [26].

2.2. Geometry

The computational domain to analyze the flow field of coughing is selected as a 2-D squared shape domain of 2m×2m size as shown in Fig 3. Further as turbulent jets are symmetric, the simulations were conducted on 1m×2m of rectangular domain to save computational time and later on the results have been plotted on full domain. As per Gupta et al [12] the nozzle depicting human mouth is considered of 0.022 m for full domain as shown in Fig.3, where nozzle is termed as inlet.

2.3. Meshing

A structured quadrilateral mesh is selected as shown in Fig.4. For selecting a suitable number of mesh elements, a Grid Independence Test (GIT) as shown in Fig. 5 is conducted. The center line velocity along the length of the computational domain is calculated after various iterations on the geometry having mesh elements in the range $6.18\times10^4$ to $8.88\times10^5$. The Figure reveals that after $1.25\times10^5$
elements the value of centerline velocity does not change for all number of mesh elements \(>1.25 \times 10^5\). Thus, geometry with the mesh of \(1.25 \times 10^5\) elements is considered for all of the current studies.

2.4. Boundary conditions

All BCs are mentioned on the meshed geometry as shown in Fig.3. The inlet boundary condition is taken as velocity inlet, all the walls are with no slip walls and outlet is taken as pressure outlet in particular the atmospheric pressure outlet.

![Figure 3 Computational domain of coughing flow field analysis](image1.png)

![Figure 4 Structured quadrilateral mesh](image2.png)

![Figure 5 Velocity profile along x-axis (centerline) of various mesh having different number of elements](image3.png)

![Figure 5 Velocity profile along x-axis (centerline) of various mesh having different number of elements](image4.png)
2.5. Parameters studied

The most important physical quantity in these kinds of studies is the coughing velocity. Based on the velocity ranges mentioned in Literature [12-14] for the real-time coughing velocities of a human, the inlet velocities to the nozzle representing human mouth are varied in the range of 6 m/s to 15 m/s on PODs of various dimensions ranging from 0.3×0.3 m to 0.3×1 m.

3. Results and discussions

The aim of the study is to find a suitable design for the geometry of a POD which can contain the infections within itself. Thus, at the outset CFD simulations are run for coughing flow field analysis to ascertain the effective range of cough in all directions to choose a suitable length of the POD. Thereafter, CFD studies are conducted by placing different shape and size PODS in the flow field of coughing jets to find the optimum POD geometry which will transmit nil infections outside of the POD.

3.1. Cough flow field analysis without pod

First, to evaluate the zone where coughing will leave its effect the simulations were conducted for different coughing velocities, and the results have been plotted in Fig. 6. Here we simulated the velocity inlet as continuous uniform velocity rather than the impulsive one as occurs in real-time coughing. But this will not affect the quality of PODS, as in this case the PODS will be overprotected and simplification of uniform velocity inlet saves lot of computational resources.

The analysis of Fig.6 indicates that even at the lowest coughing velocity of 3 m/s the effect goes far beyond 2 m from the source. Thus, these preliminary studies suggest that maintaining social distancing of 2m is not safe as coughing can travel beyond 2m which is also supported by the studies.
performed by earlier researchers [14, 27-28]. To display this phenomenon velocity contours at 6 m/s of coughing velocity have been depicted in Fig.7.

![Figure 7 Velocity contour of Cough flow field](image7)

![Figure 8 Schematic view of POD (POD-1)](image8)

Now to contain the effect of coughing to a limited zone, it has been planned to provide PODS around each human sitting in laboratories/ mess etc. To decide the dimension of the PODS a further analysis on PODS of different shape and size has been conducted in subsequent sections at an average coughing velocity of 6 m/s.

3.2. Interaction of PODS with cough jet flow field

3.2.1 Design Methodology of PODS

A design of the PODS is done using the CFD of cough jet flow field and there exists no standard POD design in literature so far. Thus, it has been decided to do simulations using CFD for a number of PODS (total 23) having different shapes and sizes. In subsequent sections analysis of each such POD is being given so that an optimum POD can be selected. The common aspect in all of the PODS is that they all have one side open to atmosphere. This will be the side where arrangements can be made for sitting of persons and ultimately the coughing jet is expected to leave from this side to the open atmosphere if the jet carries reasonably high velocities in them. The basic common geometrical details of the POD are as given in Fig.8. A simple rectangular pod is selected (as shown in Fig.8) initially for studying its effect on cough flow field. Based on the results of the study, other pods (mentioned in Figs. 9 to 36) are selected by doing further modifications in shape and dimensions of the rectangular POD shown in Fig.8. The detailed geometry shall be explained in subsequent sections for better readership along with the related discussions. Although the exit and arrangements for human sitting shall be common in all the POD geometries used in the study, the inner geometry shall be varied to get almost zero velocity at the exit of the POD for containing the infectious coughing/air jets.

3.2.2 Effect of PODS on cough flow field

Initially the POD-1 selected for the study is of cuboidal shape whose 2-D side view is rectangular in shape as shown in Fig.8. The selected pod is having a horizontal length of 0.3m and the vertical height
of 0.6m. The velocity contours for flow through this POD are shown in Fig.9. The contour clearly shows substantial redistribution of coughing jet flows. Fig.9 also shows that the exit velocity from the PODs are very high. These velocities have been drawn with respect to the vertical dimension of the POD in Fig. 13. The Figure show that the maximum exit velocity is 2.85 m/s at around 29.3 cm height from the human mouth emanating coughing jet. Thus, the above-mentioned pod is not favorable for use.

POD-1 design shows that that outlet velocity is not low so next POD-2 model is checked having dimensions of length 0.4 m and height of 0.8 m. The computational analysis of above POD-2 shows that the maximum outlet velocity is 2.43 m/s at a vertical height of 0.384 m from x-axis (Fig.13). This also shows that further change in dimension of pod is needed so as to reduce the outlet velocity. The velocity contours of the above-mentioned pod is shown in Fig.11 also confirm the same observations.

Similarly, two other PODs (3 & 4) having dimension 0.5 m ×0.6 m and 0.8m×0.6 m (Figs. 11 & 12 respectively) are studied. The reduced velocities achieved in both of the PODS are shown in Fig13. This shows that outlet velocity from POD-4 is 0.752 m/s.
Another POD-5 (Fig.14) is selected in a way having dimensions of 1m length and 0.6 m height. The computational results of this POD show a substantial reduction in outlet velocity to 0.529 m/s. Further increment in length of the POD (POD-6 &7, Figs.15-16) shows a little reduction in velocity, the above mentioned in fact outlet velocity increases with POD-7 which has 1.25m length (Figs. 16). Therefore, it has been decided to select the length of the required POD to be of 1 m.
Now, to optimize the height of the POD, the height of POD is changed to 0.8 m and 0.6 m as shown in Fig. 14 & 17 for PODS 5 & 8 respectively. The maximum outlet velocities are found to be 0.8 m/s and 0.6 m/s respectively (Fig.18) which shows that increase in height more than 0.6 m is not that desirable.

Further, it is decided to see the effect of change in shape of the PODs. Various shapes of the PODs which we considered in the study are shown in Figs 19-23 in form of PODs having diverging

Figure 18 Velocity profile of pod 5-8. Pod 5, 6 and 7 coincide on each other

Figure 19 Velocity Contour of pod 9

Figure 20 Velocity contour of pod 10

Figure 21 velocity contour of pod 11

Figure 22 velocity contour of pod 12
section towards outlet. Height at the outlet of the POD is kept to be constant at 0.6 m and at the opposite site varied from 0.1 to 0.5 m (POD 9-13) to look like a trapezium. Two more PODs (POD 14-15) are also used for the study by keeping opposite side constant at 0.6m and at outlet varies 0.8m and 0.4m respectively (Figs.24, 25 respectively). The analysis of Fig. 26 shows that the reduced outlet velocities achieved are in the range of 0.8 to 0.5 m/s. Which means that this kind of trapezium shapes considered could reduce the outlet velocity maximum to 0.5 m/s.

To see the effect of other shapes, studies are conducted on PODs 16-18 by using the combination of rectangular and circular shapes as shown in Figs 27-29 where the wall side of the PODs are replaced from rectangular to circular arc. Analysis of Fig. 30 shows that the outlet velocities achieved are again in the range of 0.8-0.5 m/s.
At the end we tried with a POD-19 having a very peculiar shape as shown in Fig.31, whose dimensions are rectangular of length 1m and height 0.6 m over which the triangle of base 0.6 m and height 0.2m is mounted, and outlet velocity is found to be close to 0.5m/s.

The velocity contours as shown in Fig.31 confirms that with changing the shape of the pod, position of maximum outlet velocity changes and thus to fix the position of outlet velocity and further reduction in its value the POD-19 has been modified to the form as shown in Figs 32-33 and Figs. 35-36. For further analysis POD shown in Fig.32 is selected and an arc, whose center is at the midpoint of height of triangle of POD-20 and radius of 0.1m is placed. Analysis shows the good reduction in outlet velocity of value 0.354m/s at the height of 0.152 m from mouth piece jet, which is very less and very close to be considered as negligible. Since from POD-20, good result could be obtained, therefore to check whether by increasing the arc length further reduces the outlet velocity the arc dimension is increased by increasing arc length as shown in Fig.33 for POD-21. The plot of outlet velocity in Fig.34 shows the maximum outlet velocity of 0.425m/s at 0.232m vertical from x-axis, thus further increasing the arc length is not a good idea. Again, to check the reduced arc length effect as shown in Fig.35 for POD-22, computational study of the above POD shows the maximum outlet velocity 0.362m/s at 0.121m vertically from x-axis (Fig.34). This is again slightly higher that the case of POD-20.
Further to check whether the increase in number of arcs at both side of the POD changes the outlet velocity favorably or not another POD-23 as shown in Fig.36 is made. The outlet velocity results obtained and drawn in Fig.34 shows that the outlet velocity increases to 0.878 m/s. This shows that it is also not a good idea to increase the number of arcs in the POD. From the above studies a conclusion can be drawn that POD-20 is best among all the studied pods. In current study most common shape are taken for pods by taking into account that fabrication of these pods will be easy and cost effective.

4. Conclusion

From the above study, it is clearly visible that maintaining 2m social distancing is not so effective in controlling the spread of SARS disease. PODs can become an effective tool for controlling the spread of SARS disease as it not only contains the coughing flows but also allows the outlet escaping air velocity to be minimum as close as to zero (0.3 m/s). POD-20 is predicted to be the most efficient POD to contain the infections due to coughing in laboratories, mess etc.

Further, as the predicted dimensions of the POD are compact, it shall require less space than social distancing conveyed by various government agencies including WHO. Therefore, it will make the fight of the space designers of laboratories, mess etc. in view of infectious diseases like COVID easier in a limited available space. Last but not the least that sanitization of these PODS will be easier and cheaper due to less enclosed space of PODS.
References

[1] Gorbalenya AE, Baker SC, Baric R, Groot RJ, Drosten C, Gulyaeva AA, Haagmans BL, Lauber C, Leontovich AM, Neuman BW, Penzar D. Severe acute respiratory syndrome-related coronavirus: The species and its viruses—a statement of the Coronavirus Study Group.

[2] World Health Organization. World Health Organization coronavirus disease 2019 (COVID-19) situation report.

[3] Hinds WC. Aerosol technology: properties, behavior, and measurement of airborne particles. John Wiley & Sons; 1999 Jan 19.

[4] Zhao B, Zhang Z, Li X. Numerical study of the transport of droplets or particles generated by respiratory system indoors. Building and Environment. 2005 Aug 1;40(8):1032-9.

[5] Morawska L. Droplet fate in indoor environments, or can we prevent the spread of infection? Indoor air. 2006;16(5):335-47.

[6] Stelzer-Braid S, Oliver BG, Blazey AJ, Argent E, Newsome TP, Rawlinson WD, Tovey ER. Exhalation of respiratory viruses by breathing, coughing, and talking. Journal of medical virology. 2009 Sep;81(9):1674-9.

[7] Lindsley WG, Blachere FM, Thewlis RE, Vishnu A, Davis KA, Cao G, Palmer JE, Clark KE, Fisher MA, Khakoo R, Beezhold DH. Measurements of airborne influenza virus in aerosol particles from human coughs. PloS one. 2010 Nov 30;5(11): e15100.

[8] Lindsley WG, Blachere FM, Beezhold DH, Thewlis RE, Noorbakhsh B, Othumpangat S, Goldsmith WT, McMillen CM, Andrew ME, Burrell CN, Noti JD. Viable influenza A virus in airborne particles expelled during coughs versus exhalations. Influenza and other respiratory viruses. 2016 Sep;10(5):404-13.

[9] Gao N, Niu J. Transient CFD simulation of the respiration process and inter-person exposure assessment. Building and Environment. 2006 Sep 1;41(9):1214-22.

[10] Bi R. A numerical investigation of human cough jet development and droplet dispersion.

[11] Badeau A, Afshari A, Goldsmith T, Frazer D. Preliminary prediction of flow and particulate concentration produced from normal human cough dispersion. InProceedings of the Second Joint 24th Annual Conference and the Annual Fall Meeting of the Biomedical Engineering Society] [Engineering in Medicine and Biology 2002 Oct 23 (Vol. 1, pp. 246-247). IEEE.

[12] Gupta JK, Lin CH, Chen Q. Flow dynamics and characterization of a cough. Indoor air. 2009 Dec;19(6):517-25.

[13] Aliabadi AA, Rogak SN, Green SI, Bartlett KH. CFD simulation of human coughs and sneezes: a study in droplet dispersion, heat, and mass transfer. In ASME International Mechanical Engineering Congress and Exposition 2010 Jan 1 (Vol. 44441, pp. 1051-1060).

[14] Thatiparti DS, Ghia U, Mead KR. Computational fluid dynamics study on the influence of an alternate ventilation configuration on the possible flow path of infectious cough aerosols in a mock airborne infection isolation room. Science and technology for the built environment. 2017 Feb 17;23(2):355-66.

[15] Zohdi TI. Modeling and simulation of the infection zone from a cough. Computational Mechanics. 2020 Oct;66(4):1025-34.

[16] Mohamed AF. Experimental measurements of far field cough airflows produced by healthy and influenza-infected human subjects.

[17] Abdel-Rahman A. A review of effects of initial and boundary conditions on turbulent jets. WSEAS transactions on Fluid Mechanics. 2010;4(5):257-75.

[18] Wan MP, Chao CY, Ng YD, Sze to GN, Yu WC. Dispersion of expiratory droplets in a general hospital ward with ceiling mixing type mechanical ventilation system. Aerosol Science and Technology. 2007 Apr 1;41(3):244-58.

[19] Deevy M, Sinai Y, Everitt P, Voigt L, Gobeau N. Modelling the effect of an occupant on displacement ventilation with computational fluid dynamics. Energy and Buildings. 2008 Jan 1;40(3):255-64.

[20] Gao N, Niu J. Transient CFD simulation of the respiration process and inter-person exposure assessment. Building and Environment. 2006 Sep 1;41(9):1214-22.

[21] Qian H, Li Y, Nielsen PV, Huang X. Spatial distribution of infection risk of SARS transmission in
a hospital ward. Building and Environment. 2009 Aug 1;44(8):1651-8.

[22] Qian H, Li Y, Nielsen PV, Hyldgaard CE. Dispersion of exhalation pollutants in a two-bed hospital ward with a downward ventilation system. Building and Environment. 2008 Mar 1;43(3):344-54.

[23] Zhao B, Zhang Z, Li X. Numerical study of the transport of droplets or particles generated by respiratory system indoors. Building and Environment. 2005 Aug 1;40(8):1032-9.

[24] Lai AC, Cheng YC. Study of expiratory droplet dispersion and transport using a new Eulerian modeling approach. Atmospheric Environment. 2007 Nov 1;41(35):7473-84.

[25] Tian ZF, Tu JY, Yeoh GH. CFD studies of indoor airflow and contaminant particle transportation. Particulate Science and Technology. 2007 Dec 5;25(6):555-70.

[26] Andersson B, Andersson R, Håkansson L, Mortensen M, Sudiyo R, Van Wachem B. Computational fluid dynamics for engineers. Cambridge university press; 2011 Dec 22.

[27] Blocken B, Malizia F, Van Drunen T, Marchal T. Towards aerodynamically equivalent COVID19 1.5 m social distancing for walking and running. Preprint. 2020 Aug 17.

[28] Pendar MR, Páscoa JC. Numerical modeling of the distribution of virus carrying saliva droplets during sneeze and cough. Physics of Fluids. 2020 Aug 1;32(8):083305.