Annals of R.S.C.B., ISSN:1583-6258, Vol. 25, Issue 2, 2021, Pages. 3998 - 4004
Received 20 January 2021; Accepted 08 February 2021.

A CFD Study on Hydraulic and Disinfection Efficiencies of the Body Sterilization Chamber

Thanh-Long Le[1,2,3*], Nguyen Tan Tien[2,3]

[1]Faculty of Mechanical Engineering, Ho Chi Minh City University of Technology (HCMUT), 268 Ly ThuongKiet, District 10, Ho Chi Minh City, Vietnam
[2]National Key Laboratory of Digital Control and System Engineering (DCSELab), HCMUT, 268 Ly ThuongKiet, District 10, Ho Chi Minh City, Vietnam
[3]Vietnam National University Ho Chi Minh City, Linh Trung Ward, Thu Duc District, Ho Chi Minh City, Vietnam

Authors Email: ltlong@hcmut.edu.vn, nttien@hcmut.edu.vn

Abstract –In this study, the numerical computation of the body sterilization chamber is used to develop a physical model and obtain the bactericidal effect of this device. It is needed to design a disinfection chamber system to server people working in areas affected by COVID-19, also ensure safety and limit the spread of this pandemic. This paper gives an illustration about using computational fluid dynamics approach to investigate the movement of disinfection solution flow, which is called anolyte. The results obtained from the numerical simulation demonstrate some of hydrodynamic values, streamlines and density of the anolyte solution. The ability to predict the hydraulic and disinfection efficiencies of this sterilization chamber is essential for obtaining the topology optimization design.

Keywords- Computational Fluid Dynamics, Disinfection Chamber, Hydrodynamic, Covid-19.

1. Introduction

Infection diseases have wreaked havoc on human communities since ancient times. As human civilization grows, so do more and more diseases. As more and more routes of trade and exchange are opened, the path of infection disease is increasingly spreading, creating first global pandemics. The plague of Justinian, The Black Death, The Great Plague of London, Smallpox pandemic …had killed hundreds of millions of people. Facing with the severe consequences of pandemics, people have found many ways to overcome them like isolating patients, developing vaccines. [1-2].

In 2020, the SARS-CoV-2 has spread across most of the world, damaging economies, claiming lives, the void created by the lack of information about the virus has rapidly driven innovation and the increasing number of human disinfection chamber is an example of rapid innovation to face with the COVID-19 global pandemic [3].Disinfection chamberworks on the principle of using an ionized salt solution in the form of mist spraying the whole body to quickly disinfect the body surface. Therefore, the most important stage in designing COVID-19 disinfection chamber is to investigate the movement of disinfection solution and then improve in the design of the chamber.

As the advancement of computational models, the potential of disinfection chamber can be assessed. Computational fluid dynamics (CFD) models are set up to simulate the movement of disinfection solution. The results demonstrate hydrodynamics values, streamlines and density of solution. The CFD method is then improved with approriate models selected, describing the

Annals of R.S.C.B., ISSN:1583-6258, Vol. 25, Issue 2, 2021, Pages. 3998 - 4004
Received 20 January 2021; Accepted 08 February 2021.

adhesion of saline solution to the human bobdy, pathogen inactivation.Computational models are invaluable tools for simulating disinfection processes as they can reproduce the conditions encountered experimentally. The optimization of hydraulic efficiency will facilitate more uniform contact time as well as increase in the level of pathogens inactivation [4].

2. Methodology

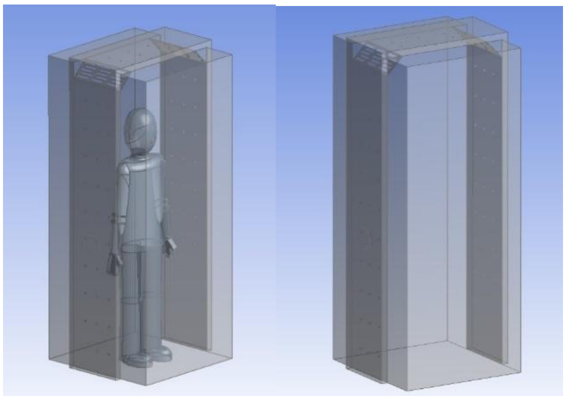
A. Physical model

In this study, a physical model of the sterilization chamber is designed which represents all important features of this chamber in real life. The model is ensured that it could be widely used and suitable for almost all the Vietnamese people who have the average height are 1.65 meter. Because of those demands, this chamber has a 0.8m by 1m floor area and a 2m high ceiling which about the size of a single-person. This disinfection room has one entry point and one exit point, a chemical tank, a spray mechanism system. According to the size of the chamber, the fluid flow is calculated to make sure that the disinfection fluid flow is sprayed all over the room after 20 seconds.

$$q = \frac{\delta V}{\delta t} = \frac{1,6}{20} = 0,08\, m^3/s \qquad (1)$$

The liquid that is used for this process is anolyte solution (Anolyte ClO). In order to cover all of the human body and the walls of the chamber, the solution is transfer into dew by using ultrasonic nozzles. These nozzles are located along the walls and can create dew of which diameter from 1 to 5μm.

The model was built based on the sterilization chamber in real life. However, the main purpose of this study is to observe the distribution of the solution inside the room, it means



some solid parts of this chamber are not included for simplifying the computation (Figure 1).

Figure 1. A model of the disinfection chamber

In the design of sterilization chamber, the solution is sprayed through the nozzels which are installed around inside of the room. Based on the purpose of this study, obtaining the simulation's results, the Navier-Stokes equation and some relevant equations are used to analyzed the fluid flow in 3-D dimension.

Navier-Stokes equation:

$$\rho\left(\frac{\partial \mathbf{u}}{\partial \mathrm{t}}+\mathbf{u}.\nabla\mathbf{u}\right)=-\nabla \mathrm{p}+\nabla.\mathbb{T}+\mathbf{F} \qquad (2)$$

The conservation of mass:

$$\frac{\partial\rho}{\partial t}+\nabla.(\rho.\mathbf{u})=0 \qquad (3)$$

where:

$\rho$is the density(SI unit: kg/m^3)

uis the velocity vector (SI unit: m/s)

p is the pressure (SI unit: Pa)

F is the volume force vector (SI unit: N/m^3)

In this study, sterilization chamber is analyzed by the realizable k-ε module. According to T.-H Shin et al [10], this module is widely known for calculating turbulent flow by evaluating kinetic enery equation (k) and turbulent dissipation equation (ε). Due to the fact that it can obtained better results in turbulence and reduce the number of calculations, the realizable k-ε module is often used for turbulent model in study or industrial applications[10-11].

The kinetic enery equation (k):

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_j}(\rho k u_j)$$
$$=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+P_k+P_b-\rho\epsilon-Y_M+S_k \ (4)$$

The turbulent dissipation equation (ε):

$$\frac{\partial}{\partial t}(\rho \epsilon)+\frac{\partial}{\partial x_j}(\rho \epsilon u_j)$$
$$=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right]+\rho C_1 S_\epsilon-\rho C_2\frac{\epsilon^2}{k+\sqrt{\nu\epsilon}}+C_{1\epsilon}\frac{\epsilon}{k}C_{3\epsilon}P_b+S_\epsilon \qquad (5)$$

where:

$$C_1=max\left[0.43,\frac{\eta}{\eta+5}\right]$$
$$\eta=S\frac{k}{\epsilon}$$
$$S=\sqrt{2S_{ij}S_{ij}}$$

u_j is velocity component in corresponding direction

$\mu_t$is eddy viscosity, $\mu_t=\rho C_\mu\frac{k^2}{\epsilon}$

$C_\mu=0.09$,$\sigma_k=1$

$\sigma_\epsilon=1.3$, $C_{1\epsilon}=1.44$.

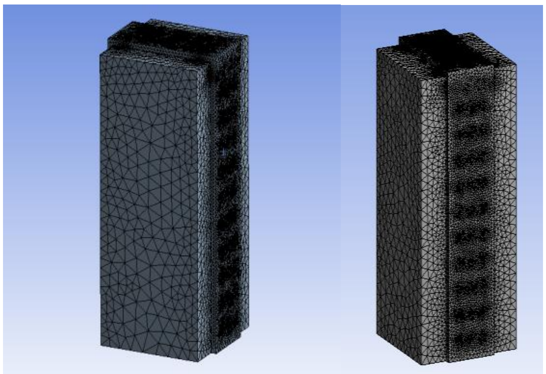
All of the surface, including the human body is set as walls, the inlets are located in the ultrasonic nozzles which have the speed at 2m/s. The orientation of this flow is perpendicular to the inlet.

Annals of R.S.C.B., ISSN:1583-6258, Vol. 25, Issue 2, 2021, Pages. 3998 - 4004
Received 20 January 2021; Accepted 08 February 2021.

B. Numerical method

Due to the development of computer technology, Computational Fluid Dynamics now is widely used to estimate the movement and hydrodynamic characteristics of an object in fluid enviroment. Most research is based on the Navier-Stokes equation and initial boudary conditions to analyze and obtain the velocity values, pressure distributions, kinetic energy, etc [5]. The continuity, momentum and energy equations areused to analyze all type of fluid flow in the present study. Since it is too difficult to calculate all of hydrodynamic values by using analytical method to solve these equations, CFD is applied to figure out the results. Nevertheless, estimating values by CFD is help to rise the accuracy and reduce the calculation time[6]. The studies of M. Feurhuber el at [7-9] show that is appropriate to use CFD in prediction ofhydrodynamic characteristics and movement of fluid flow inside of sterilization chamber.This study used a combination of Computational Fluid Dynamics (CFD) and Fluent module of Ansys Workbench to give predictions about the behavior and movement of disinfection solution in this sterilization chamber.

After having the physical model, the meshing is generated. In the first case that nobody inside the chamber, there are 363657 nodes and 1946212 elements is used. The other case



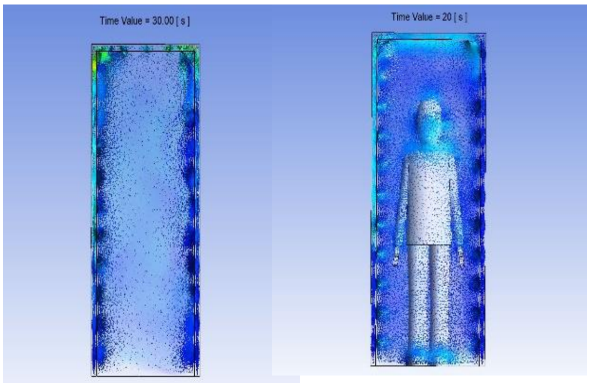
which has one person in the room has 449018 nodes and 2392611 elements in total (Figure 2).

Figure 2. Typical mesh used in the computational domain for the disinfection chamber

3. Results and Discussion

Figure 3illustrate the distribution of the solution flow in 20 seconds after the operation. The results obtained from the simulations showed that the flow is spreaded out all over the chamber and the human body. The face and hands areas are fill with dense solution flow, which is good since these parts could be infected easily with the virus. Figures4 and 5 give demonstrations about the streamline of the solution flow and its velocity during the operation. The maximum speed (2m/s) occurs at the nozzels and the minimum speed appear at the bottom of the chamber.

Annals of R.S.C.B., ISSN:1583-6258, Vol. 25, Issue 2, 2021, Pages. 3998 - 4004
Received 20 January 2021; Accepted 08 February 2021.



(a) (b)

Figure 3. The flow distributions of the chamber (a) Without human; (b) With human

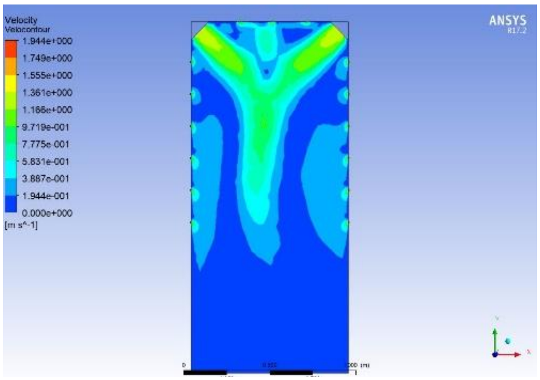


Figure 4. The velocity field of the chamber.

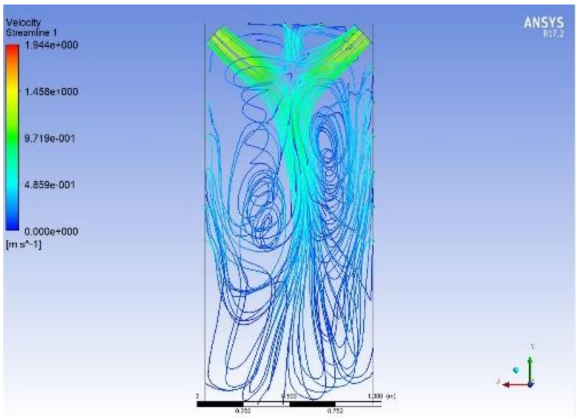


Figure 5. The streamlines of the chamber.

Based on Figure 5, the distributione of flow is approriate since it covers all of the space in sterilization chamber. This information can be useful in further designs which have to optimize the design, set the suitable location for the nozzels and reduce the number of these nozzels.

Figure 6 show the pressure in the chamber during 20 seconds period. This pressure field is uniformly distributed and hit the peak at the upper of the chamber. In the first few seconds, the value of pressure is little chaos. There is a huge distance between highest and the lowest presssure values during the three-seconds.After this period, the air flow moves from a high pressure region to a low one and reduce the diference between the top and the bottom values. This phenominon is described in Figures 7 and 8.

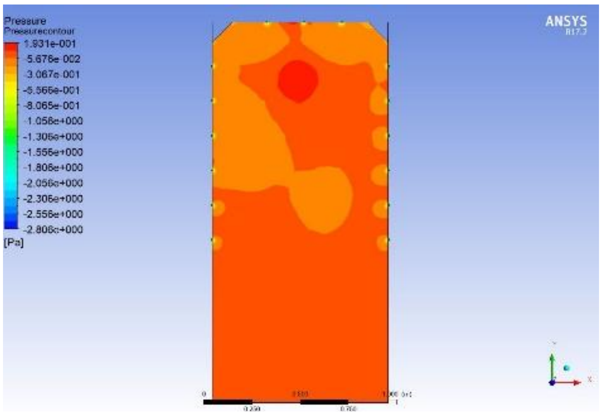


Figure 6. The pressure field of the chamber.

However, the general trend of pressure value of chamber which has human inside is different consider to the empty one. In Figure 7, due to the human body, the area in the chamber is compilcated and narrow. It means the airflow will be harder to move from high pressure area to low pressure region. As a consequence, the maximum and minimum lines in Figure 7 is slightly decreased since it takes time for the pressure value to goes down. Because of the emptiness, the chamber which do not have anything inside is easier for the airflow to transfer. As we can see in Figure 8, the maximum and the minimum value are stable except for the five-seconds period at the beginning.

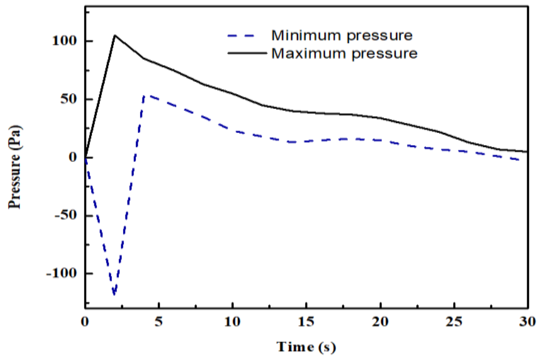


Figure 7. The maximum and minimum value of pressure inside the chamber in 30 seconds (with human)

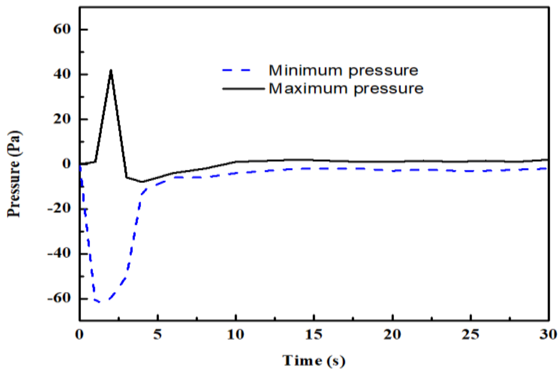


Figure 8. The maximum and minimum value of pressure inside the chamber in 30 seconds (without human)

4. Conclusion

In this study, an sterilization chamber concept design has been modelled, meshed and analysed to obtain the characteristics such as pressure and velocity distribution, streamlines, etc. The figures obtained from the numerical results showed that this chamber is suitable to put into practice. It also gives potential to upgrade and optimal the new designs in further studies.

Acknowledgment

This research is supported by DCSELAB and funded by Vietnam National University Ho Chi Minh City (VNU-HCM) under grant number TX2021-20b-01. We acknowledge the support of time and facilities from Ho Chi Minh City University of Technology (HCMUT), VNU-HCM for this study.

References

[1]. Wickramatillake, C.Kurukularatne, SARS-CoV-2 human disinfection chamber: A Critical Analysis (2020), Occupational Medicine, 1-2.

[2]. S. Haensch, R. Bianucci, M. Signoli, M. Rajerison, M. Schultz, S. Kacki, M. Vermunt, D. A. Weston, D. Hurst, M. Achtman, E.Carniel, B. Bramanti, Distinct Clones of Yersinia pestis Caused the Black Death(2010).

[3]. S. Riedel, E. Jenner, the History of Smallpox and Vaccination, Baylor University Medical Center Proceedings, 18:1, 21-25.

[4]. A.Angeloudis, Numerical and Experimental modelling of flow and kinetic processes in serpentine disinfection tanks, Cardiff School of Engineering (2014).

[5]. P.G. Marshallsay, Use of computational Fluid Dynamics as a Tool to Assess the Hydrodynamic Performance of a Submarine, 18th Australasian Fluid Mechanics Conference (2012).

[6]. P. K. Kundu, I. M. Cohen, D. R. Dowling, Fluid Mechanics (2012), 421-472

[7]. M. Feurhuber, A. Cattide, M. Magno, M. Miranda, R. Prieler, C. Hochenauer, Prediction of the fluid flow, heat transfer and inactivation of microorganism at medical devices in modern steam sterilizers using computational fluid dynamics, Appl. Therm. Eng. 127 (2017), 1391–1403.

[8]. M. Feurhuber, M. Magno, M. Miranda, C. Hochenauer, CFD investigations of steam penetration, air-removal and condensation inside hollow loads and cavities, Appl. Therm. Eng. 147 (2019), 1070–1082.

[9]. M. Feurhuber, M. Magno, M. Miranda, R. Prieler, C. Hochenauer, CFD investigation of non-condensable gases in vacuum and non-vacuum steam sterilizers, Chem. Ing. Tech. 91 (4) (2019), 1–13.

[10]. T-H. Shih, W.W. Liou, A. Shabbir, Z. Yang, J. Zhu, A new k-ε eddy viscosity model for high Reynolds number turbulent flows (1995), Computers & Fluids, 227-238.

[11]. ANSYS Fluent User's Guide.