MASTER'S THESIS REVIEW

Author: **Nalugotla Elijah Might**
Title: **CFD Model and Simulation for the Design of a Spray Booth**
Supervisor: **Ing. Martin Barták, Ph.D.**
Reviewer: **Ing. Petr Zelenský**

Master's Thesis assignment

According to the assignment of the Master's Thesis, the students was targeting the issue of proper air flow in the industrial spray booth for spray painting. He was assigned to create a numerical model to simulate air flow in a booth with a working chamber of approx. 1 m^3 volume and to perform CFD simulation in order to assess different variants of the exhaust opening(s) design and location, with respect to the uniformity of air flow in the working chamber of the booth. Also, he was supposed to assess the effectiveness of paint particles transport to the filtration system.

I consider the topic relevant for extending the theoretical and practical knowledge about airflow simulation in HVAC systems and devices. It also extends the current knowledge on spray booths design, possibly resulting in a design optimization. The chosen method (CFD modelling and simulation) is suitable for solution of the investigated issue. It puts adequate demands on the student aspirin to graduate in the Master's study program at the Faculty of Mechanical Engineering, CTU in Prague.

Master's thesis content

The submitted Master`s thesis has 78 pages in total. In the introduction part of the thesis, the author briefly presents the issue of spray painting, including its advantages and disadvantages. In relation, the option to use spray booths is mentioned in general. The basic objectives of the work are defined, i.e. to simulate various options of the portable spray booth exhaust outlet positioning using ANSYS Fluent 19 as the software for CFD simulations. The main target of the master's thesis was an optimization of the spray booth design in order to achieve a uniform air flow in the working chamber of the booth.

The following chapter is summarizes the research background and current state-of-the-art of the researched issue. Specifically, (i) the issue of spray painting, (ii) the standard designs of spray booths considering methods of collecting the overspray and direction of the air flow in the working chamber, (iii) the application of CFD method for simulation of air flow in the HVAC devices and the work flow of CFD studies from numerical model preparation to results evaluation. The methods of turbulence approximation, modelling of pressure-restrictive devices and ways of particle transfer simulation is described in detail.

Chapter 3 presents the design of the spray booth and introduces simplification of its geometry necessary for the CFD simulation. Several variants of the exhaust openings location are proposed. The author intended to simulate 6 design variants of exhaust opening positioning (various locations at the bottom, back and top walls of the spray booth) and 3 design variants of front inlet opening (fully opened inlet and 2 variants of partially opened inlet). The filters which are used in the spray booth are introduced. The ways of their numerical modelling using PorZo tool is discussed as well.

Chapters 4 and 5 are targeting the preparatory steps of the CFD simulations, namely meshing of the numerical model and setting of the physical models and turbulence conditions. The issue of proper meshing of near-wall regions is discussed and properly addressed.

The results of the simulations are evaluated in Chapter 6. First, the author discusses the results of the simulations of the 6 design variants of the exhaust opening location, considering two inlet air flow velocities recommended for the industrial spray booths (minimum and maximum recommended inlet velocity 0.5 m/s and 2.5 m/s, respectively). The initial discussion is based on comparison of velocity vectors and velocity fields in the cross section of the spray booth. Additionally, the velocity profiles along the height of the inlet opening are evaluated and two most optimum design variants are proposed. The air flow in the spray booth with the fully opened inlet and partially closed inlet is studied as well. The last part of the chapter is targeting the simulation of particle distribution in the spray booth, using the Discrete Phase Model (DPM). Spreading of particles with two different sizes is assessed, for two most optimal design variant of the exhaust location.

The whole outcome of the research is briefly summarized at the end of the thesis. The author also states the most important knowledge and recommendations arising from the work.

The content of the work is well balanced. Approximately 60 % of the Master's thesis is dealing directly with solving of the assigned research issue. The author has demonstrated that he is able to solve the given subject individually to the extent that meets the requirements of the Master's thesis.

Comments of the reviewer on the Master's Thesis

The topic of the thesis is targeting practical issue within HVAC engineering. The author carried out a numerical variant study targeting design optimization of a spraying booth. He compared 6 design variants of exhaust location, considering two inlet air flow velocities and 2 design variants of front inlet opening, using ANSYS Fluent. He evaluated and compared results of the CFD simulations and proposed two most optimal design options.

The text of the thesis is well written and comprehensible. The individual chapters logically follow one another. The only exception is Sub-chapter 2.4 – Spray painting, which may be more appropriate to place at the very beginning of the Chapter 2 – Literature review, in order to better introduce the motivation of the research, i.e. the use of the spraying for practical applications and the adequate removal of the over-spray during industrial spraying process to protect the operator. Considering the formal side, the thesis has minor deficiencies, such as grammatical errors (mostly misspellings), typos and wrong formatting of some paragraphs (pages 6 and 10).

In the practical part of the Master's thesis, I find the part dealing with creation of the numerical mesh insufficiently comprehensive. The author mentioned in detail the approach to estimate the height of the 1st cell in the near-wall region, however he did not clearly describe the boundary layer mesh, where the quality network is particularly important (i.e. total number of cells and growth factor), neither the size of the cells in the free space. The size of the cells can be estimated from figures 4-1. to 4-3. However, it should be indicated in a more clear way, for example, in a table.

In Chapter 5, the author compared residual plots from two simulations with two different types of boundary conditions prescribed to the exhaust opening – velocity inlet and mass flow outlet. Author tested their performance during the simulation. However, there is insufficient discussion of this numerical study and no clear conclusion of the author's findings. Further on, the author used velocity inlet as a boundary condition without clear reasoning why.

The main issue to remark is the insufficient referencing of the literature. Some of the important findings in the text were taken from other publications, but the author did not mention the source in a proper way. The most serious mistake is in the reference Chang, N. K., Woo, H. C., Suk, J. C., Chang, H. P., & Dong, S. K. (2002), where the name of the reference publication is completely missing. The reference mentions only the authors and that the publication was found at on-line database Elsevier. Also, the references to web pages may include a referenced topic (issue) that is cited, at least in general. This concerns following references: CJ Coatings UK. (2018), Jicolor, s.r.o. (2018), pati. (2018) VITON, s.r.o. (2018) Vondál, J. (2018), WAGNER, GmBH (2018).

Recommendations for the defence debate

1. As it is correctly stated in the Master's Thesis, one of the most important issues related to CFD simulation is correct meshing of the numerical model. Therefore, it would be appropriate for the author to present a complete summary of the parameters of the created mesh, i.e. not only the height of the first cell by the walls, but also the characteristics of the numerical mesh in the near-wall region (first cell height, number of cells and growth rate of the boundary layer mesh) and size of the cells in the free space. Additionally, he can specify the maximum and minimum cell sizes in the model.
2. I would like to ask the author to clarify what k-ε model of turbulence did he use to solve the influence of turbulence on the air flow in the CFD simulations and what was the reason to select that particular model.
3. I would like to ask the author to briefly discuss the outcomes of the chapter 5.1 (Exhaust outlet) and better specify the findings of the numerical study (aside of the residual plots). The author may especially clarify what was the reason to use the velocity inlet boundary condition as a boundary condition of the exhaust from the spraying booth in the CFD simulations following this initial numerical study.

Overall rating

The content of the Master's Thesis is well balanced; the text is adequately accompanied with graphical outputs. Analysis of the results is logical and I agree with the conclusions. The assignment of the Master's Thesis was fulfilled. However, the author made minor mistakes. The most serious issues to remark are the insufficient referencing to the literature and insufficient description of the numerical mesh, which could make it more difficult to use the results of the thesis for further research. However, these factors should not affect the results achieved.

I propose an overall rating:

B (very good).

Prague, 6th August 2018

Ing. Petr Zelenský