October 2021

Modeling the Ambient Conditions of a Manufacturing Environment Using Computational Fluid Dynamics (CFD)

Yang Liu
*University of South Florida*

Follow this and additional works at: [https://digitalcommons.usf.edu/etd](https://digitalcommons.usf.edu/etd)

Part of the Mechanical Engineering Commons, and the Sustainability Commons

Scholar Commons Citation

Liu, Yang, "Modeling the Ambient Conditions of a Manufacturing Environment Using Computational Fluid Dynamics (CFD)" (2021). *USF Tampa Graduate Theses and Dissertations.*
https://digitalcommons.usf.edu/etd/9694

This Thesis is brought to you for free and open access by the USF Graduate Theses and Dissertations at Digital Commons @ University of South Florida. It has been accepted for inclusion in USF Tampa Graduate Theses and Dissertations by an authorized administrator of Digital Commons @ University of South Florida. For more information, please contact scholarcommons@usf.edu.
Modeling the Ambient Conditions of a Manufacturing Environment Using Computational Fluid Dynamics (CFD) 

by 

Yang Liu 

A thesis submitted in partial fulfillment of the requirements for the degree of Master of Science in Mechanical Engineering
Department of Mechanical Engineering
College of Engineering
University of South Florida

Major Professor: Nancy Diaz-Elsayed, Ph.D.
Susana Lai-Yuen, Ph.D.
Wenbin Mao, Ph.D

Date of Approval:
October 25, 2021

Keywords: CFD Simulation, Digital Twin, Smart Manufacturing, Green Manufacturing

Copyright © 2021, Yang Liu
Acknowledgments

I'd like to thank my professor Dr. Nancy Diaz-Elsayed for her patience and attentive guidance and always encouraging me to complete higher challenges. Her timely help has allowed me to accomplish this thesis better and has facilitated my growth as a researcher. Her seriousness and rigor in academics are a lifelong role model for me to follow. I am fortunate to have grown up in professor Dr. Nancy Diaz-Elsayed's team.

Also, I would like to thank my thesis committee team. Dr. Susana Lai-Yuen for giving me fruitful discussions and insights. Dr. Wenbin Mao's valuable comments on my thesis enabled me to improve its quality.

I would also like to thank everyone in the S3 laboratory. Mr. Sohan Nagaraj for giving me a deeper understanding of CNC machines. Ms. Purvee Bhatia and Mr. Bharat Pulaparthi for helping me from thesis to defense. And Mr. Donald Mccleary, Mr. Scott Bartlett, Mr. Giang Dang, and Mr. Muhammad Saad Amjad for their advice on my thesis paper.

I would also like to thank my family for their support during this year of study. My wife, Ms. Kaiwen Lin, spends more time caring for two kids to have enough time to do my research and for the love and support, she gave me every minute of this journey. My father, Mr. Tiping Liu, and my mother, Ms. Hong Jiao, have encouraged me and help make my research path easier. My two lovely babies are very understanding and do not bother me when I am working.
# Table of Contents

List of Tables .................................................................................................................. iii

List of Figures ................................................................................................................... v

Abstract ............................................................................................................................. viii

Chapter 1: Introduction ..................................................................................................... 1

Chapter 2: Literature Review .......................................................................................... 3
    2.1 CFD Technology ................................................................................................. 3
    2.2 CFD Applications for Ambient Environment .................................................... 3
    2.3 CFD Applications for Manufacturing ................................................................. 5
    2.4 CFD Application for Manufacturing Processes ................................................ 6
    2.5 Advantages of CFD ............................................................................................ 6
    2.6 Thermal Issues in Manufacturing ........................................................................ 8
    2.7 HVAC System and Thermal Comfort ................................................................. 8
    2.8 Digital Twin ........................................................................................................ 10
    2.9 Research Goal and Objective ............................................................................. 10

Chapter 3: Methodology .................................................................................................. 12
    3.1 Test Room and Flow Domain ............................................................................. 12
    3.2 Experimental Facilities ...................................................................................... 14
    3.3 Overview of Methodology .................................................................................. 15
    3.4 Airflow Investigation ......................................................................................... 17
        3.4.1 HVAC System in Machining Laboratory ....................................................... 17
        3.4.2 Airflow in Machining Laboratory ................................................................. 18
        3.4.3 Assumption .................................................................................................. 18
        3.4.4 Airflow Model ............................................................................................. 19
        3.4.5 Viscous Model ............................................................................................ 21
        3.4.6 Optimization Solutions for k-ε Model ......................................................... 24
    3.5 Heat Generation in the Machining Laboratory ................................................... 25
        3.5.1 Heat Generation .......................................................................................... 25
        3.5.2 Energy Equation ......................................................................................... 26
        3.5.3 Heat Transmission Method ........................................................................ 28
        3.5.4 Linear Thermal Expansion ....................................................................... 29
    3.6 Meshing Method ................................................................................................... 30
        3.6.1 Element Quality ......................................................................................... 32
        3.6.2 Aspect Ratio ................................................................................................ 34
        3.6.3 Independent Study ..................................................................................... 36
3.7 Modeling Ambient Conditions via CFD

3.7.1 CFD Simulation Scenarios

3.7.2 Scenarios Data

3.7.3 Validation

Chapter 4: CFD Results and Validation

4.1 Scenario 1: Case 1 and Case 2 CFD Results Comparison and Validation

4.2 Scenario 2: Case 3 and Case 4 CFD Results Comparison and Validation

4.3 Scenario 3: Case 5 and Case 6 CFD Results Comparison and Validation

4.4 Limitations and Discussion

Chapter 5: Conclusion and Future Research

References

Appendix A: Element Quality Table

Appendix B: Aspect Ratio Table

Appendix C: Location of Temperature Data Collection Point A to D in Machining Laboratory for Scenario 1

Appendix D: Location of Temperature Data Collection Point A to D in Machining Laboratory for Scenario 2

Appendix E: Location of Airflow Data Collection Point A to G in Machining Laboratory for Scenario 3

Appendix F: Location of Temperature Data Collection Point A to D in Machining Laboratory for Scenario 3
List of Tables

Table 1: The scope of the reviewed articles with CFD applications. ...........................................11
Table 2: Parameter of mixed thermal conditions. ...........................................................................28
Table 3: Linear expansion coefficient of different materials, at normal temperature
(20 °C)...........................................................................................................................................30
Table 4: Three scenarios. .................................................................................................................38
Table 5: Case 1 parameter of airflow speed and temperature to case 6 parameter of
airflow speed and temperature. ..................................................................................................39
Table 6: Case 1 parameter of lights, CNC machine and operator to case 6 parameter
of lights, CNC machine and operator. .......................................................................................39
Table 7: Free stream temperature for the lights, CNC machine, and operator. ....................40
Table 8: Scenario 1 CNC machine deformation under different materials. ............................45
Table 9: Temperature percentage error of case 1 and case 2. .....................................................46
Table 10: Airflow speed error of case 1 and case 2. .....................................................................48
Table 11: Temperature percentage error of case 3 and case 4. ....................................................55
Table 12: Scenario 2 CNC machine deformation under different materials. ............................58
Table 13: Airflow speed simulation data, validation data, and error for case 3 and
case 4. ............................................................................................................................................60
Table 14: Temperature percentage error of case 5 and case 6. ...................................................64
Table 15: Scenario 3 CNC machine deformation under different materials. ............................66
Table 16: CNC machine deformation under different materials between opening and
closing doors........................................................................................................................................67
Table A.1: Element quality table ..................................................................................................88
Table B.1: Aspect ratio ..........................................................................................................................90

Table C.1: Location of temperature data collection point A to D in machining laboratory for scenario 1. .................................................................................................................................92

Table D.1: Location of temperature data collection point A to D in machining laboratory for scenario 2. .................................................................................................................................93

Table E.1: Location of airflow data collection point A to G in machining laboratory for scenario 2. .................................................................................................................................94

Table F.1: Location of temperature data collection point A to D in machining laboratory for scenario 3. .................................................................................................................................95
List of Figures

Figure 1: PMV index. ...........................................................................................................................................9

Figure 2: SolidWorks model of the S3 laboratory. ..............................................................................................12

Figure 3: The S3 laboratory simulated in SolidWorks and the air domain. .........................................................13

Figure 4: A and B is the location of the Govee thermometer; point A is used to collect boundary conditions data; point B is used to verify the accuracy of the CFD simulation. .................................................................14

Figure 5: Flow chart of methodology. ..................................................................................................................16

Figure 6: SolidWorks model of the S3 laboratory. ..............................................................................................18

Figure 7: Convergence comparison of the (a) k-ε model and (b) k-ω model. .......................................................23

Figure 8: (a) CNC machine and (b) heat generated inside of CNC (c) heat generated by front side of CNC machine; (d) heat generated by the back side of CNC machine. .........................................................................................25

Figure 9: (a) Heat generation by the operator (b) heat generation by the operator light. ......................27

Figure 10: (a) Face sizing method for interior faces of machining laboratory (b) face sizing method for exterior walls of machining laboratory.................................................................31

Figure 11: (a) Tetrahedral meshing method (b) polyhedral meshing method. .....................................................33

Figure 12: (a) ANSYS Fluent numerical results of element quality (b) ANSYS Fluent graphical results of element quality. .............................................................................................................34

Figure 13: (a) ANSYS Fluent numerical results of aspect ratio (b) ANSYS Fluent graphical results of aspect ratio. ........................................................................................................................................35

Figure 14: Independent study for average and maximum outlet airflow speed..............................................36

Figure 15: Independent study for CNC machine temperature. ........................................................................37

Figure 16: (a) Temperature contour and (b) temperature volume rendering for case 1. ..................42
Figure 33: Temperature contour in case 6. ........................................................................................................64

Figure 34: (a) Temperature (294.4k to 300.0k) and (b) airflow speed (0 m/s to 4.468 m/s for case 5; (c) temperature (294.6k to 300.4 k) and (d) airflow speed (0m/s to 4.312m/s) for case 6........................................................................................................65

Figure 35: (a) 2D vertical view of contour near CNC machine for case 5; (b) 2D vertical view of contour near CNC machine for case 6........................................................................................................65

Figure 36: (a) Temperature and (b) airflow speed for case 5; and (c) temperature and (d) airflow speed for case 6. ........................................................................................................................................66

Figure 37: (a) Height and width of the CNC machine and (b) internal view of the CNC machine in coordinate system........................................................................................................................................68

Figure 38: (a) Airflow streamline in volume rendering for case 5 and (b) airflow streamline in volume rendering for case 6. ........................................................................................................................................69
Abstract

As manufacturing equipment evolve to higher speed and require high precision operations, the impact of environmental changes on machine accuracy becomes critical. Due to thermal expansion, the structure of the machine can change when ambient temperature varies. When the airflow in the laboratory changes, this also alters the operator's thermal comfort. Either the change in machine structure or operator comfort can ultimately affect machine accuracy. The manufacturing industry is currently using heating, ventilation, and air-conditioning (HVAC) systems to regulate the temperature of the working environment. However, since conventional HVAC systems determine whether to activate the HVAC system by collecting the temperature difference around the thermostats, this situation delays air activation time and misses the optimal time to change the factory workshop's internal temperature. This not only fails to ensure the accuracy of high-precision equipment, but also causes energy waste, so an accurate method of monitoring the indoor environment is beneficial to the development of green manufacturing and smart manufacturing.

This research uses computational fluid dynamics (CFD) techniques to simulate thermal and airflow distribution in three controlled experiments in the S3 laboratory at the University of South Florida. The experiments were conducted to investigate the changes in temperature and airflow throughout the laboratory influenced by the access of machine operators, computer numerical control (CNC) machine start-up and shutdown, seasonal changes, and the opening and closing of the laboratory door. This experiment is conducted to verify the accuracy of the simulation results
and investigate the effects of temperature changes and airflow changes on the thermal deformation of the CNC machine.

The average percentage error of the temperature for the three control experiments varied from 0.08% to 0.15%. The error range of the airflow velocity varied from 0 m/s to 0.65 m/s. The maximum structural change in the height of the CNC machine was found to be 0.085 mm. Even with seasonal changes, such errors show the advantages of CFD technology over conventional HVAC systems. In conclusion, CFD simulation technology can address structural changes in high-precision equipment due to temperature changes, changes in operator comfort due to changes in airflow, and has the potential to reduce energy waste with more informed control of the HVAC system.
Chapter 1: Introduction

Fifty-five percent of the world's energy consumption is attributed to manufacturing while emitting twenty-one percent of global greenhouse gas emissions [1]. The modern U.S. is a highly industrialized country, and in 2020, 36% of total U.S. end-use energy consumption came from the industrial segment [2]. The amount of electricity consumed to cool the internal temperature of buildings is approximately 392 billion kilowatt-hours (kWh) (U.S. residential and commercial sectors), equivalent to 10% of the total U.S. electricity consumption in 2020 [3]. This indicates the extensive use of HVAC systems in the modern manufacturing industry. The initial motivation for installing HVAC systems in the manufacturing industry was to provide a good working environment for operators and machines. However, as HVAC systems became popular, problems arose from the usage of HVAC systems including the waste of energy and operator discomfort from temperatures that are too high or too low; according to the Workplace (Health, Safety and Welfare) Regulations 1992, it is recommended that workplaces guarantee an indoor temperature of at least 16 °C, or 13 °C if the workplace requires the operator to perform rigorous physical effort [4].

The U.S. government has launched various policies [5] to guide manufacturing towards greener manufacturing, requiring manufacturing to improve efficiency, expand clean energy, develop sustainable manufacturing, and mobilize employees and the Department of Energy community to achieve environmentally-friendly manufacturing [6]. Also, with the coming of Industry 4.0, and the increase in automation, intelligent systems, and system complexity, there is a need for manufacturing to become more efficient and save energy.
Considering the manufacturing factory perspective, modern advances in information technology have driven manufacturing facilities toward high speed machining technology to meet the requirements of the aerospace, automotive and new energy industries [7]. This technology requires high speed and high positioning accuracy of CNC machine tool feed systems to be continuously improved. However, as the precision of CNC machines increases, the pending challenges can consist of an increase in thermal deformation errors (thermal expansion of materials), a decrease in the structural stiffness of the system, an increase in vibration, and a decrease in machine accuracy [8][9]. The manufacturing industry needs better approaches to detect and observe variations in the ambient environment. Meanwhile, conventional HVAC systems are controlled by thermostats. The thermostat does not necessarily reflect the internal temperature of the actual manufacturing workshop due to the distance of the thermostat from the workers and equipment, the complex conditions of human access in the room and the existence of many experimental equipment, which makes it infeasible for the traditional HVAC system to accurately detect and control the ambient temperature and meet the manufacturing industry requirement efficiently [10].

So, there is an urgent requirement for a new approach to detect and optimize temperature and air changes in the manufacturing industry, both at the macrocosm level of the manufacturing sector and the microcosm level of manufacturing factories.

This experiment takes advantage of CFD technology to simulate the environment inside a modern manufacturing laboratory. This experiment investigates the entry and exit of people, the start and stop of the CNC machine, seasonal changes, as well as the outdoor environment's impact on the indoor environment.
Chapter 2: Literature Review

2.1 CFD Technology

CFD is a branch of fluid mechanics established on Navier–Stokes partial differential equations and utilizes numerical analysis and data structure to solve fluid flow problems [11]. CFD simulates fluid flow by iteratively reducing the error between the approximate solution and the partial differential equation solution through numerical methods such as finite element method, finite difference method, and finite volume method [12].

2.2 CFD Applications for the Ambient Environment

Facilities in the manufacturing industry are becoming more and more sophisticated while bringing more factors that can affect the precision of production equipment. For example, carbon film deposition is influenced by residual hydrocarbons in the ambient environment surrounding the exposure chamber. Hydrocarbons are caused by water vapor in the ambient environment or by adsorbed water induced by extreme ultraviolet lithography (EUV) [13]. At the same time, the impact of ambient temperature variations on operators in the manufacturing sector reduce the output of the firm-level total factor productivity (TFP), production factor inputs, and production output [14]. The proper approach to simulating the environment is crucial because the operation of equipment in the appropriate environment can improve the precision of the machine and keep the operator comfortable in the environment, which can further advance manufacturing. For future research, this experiment needs to be informed of the simulation ambient research topics that have been done so far. For example, indoor airflow simulation to help hospitals analyze the spread of air and bacteria situations of isolated rooms [15] (especially after the covid-19 global outbreak has
practical applications) and indoor temperature simulations allow comparison of the differences in energy consumption under conventional HVAC systems and a simulated temperature-controlled HVAC system [16].

Since CFD has a unique advantage in simulating environments, CFD technology has various applications in a wide range of field-related ambient environments. From small-scale simulations such as identifying the human heat transfer coefficient [17] to large-scale simulations such as city simulation [18], prior research has shown relatively accurate results. CFD techniques have demonstrated an outstanding application for simulating the built environment, such as indoor housing, office, and poultry farming. The following are specific examples.

- **Indoor Housing**

  The harmful substances of formaldehyde and benzene compounds produced by indoor renovation can cause upper respiratory irritation symptoms, headaches, fatigue, and rash [19]. CFD numerical simulation technology predicts where the air in the room is fresh, changes the location of the air conditioner, and improves the thermal comfort of people in the room. The advantages can save manpower and material resources [20].

  CFD technology has also been used to simulate the ventilation of high-rise interiors under different conditions of closing and opening doors, providing builders with more rational planning and design [21]. In this case, CFD technology is able to reduce energy consumption

- **Office**

  CFD simulation techniques have been coupled with building energy models to optimize non-uniform thermal environments due to varying air supply in large office areas [22]. The experiment uses a stable two-way coupling and the EnergyPlus model outputs with the surface
wall temperatures and subzone airflow rates to the CFD program. The CFD simulation exports the air mass exchange rates to the EnergyPlus simulation.

The program given by CFD simulation can save 3.5% of cooling energy and propose a practical approach to solve large open areas in the office, where occupants complain about feeling too cold or too warm.

- **Poultry Farming**

  CFD technology was also used for poultry farming. Poultry breeding housing is naturally ventilated, and the effectiveness of ventilation determines the yield of the farmer. Reasonable planning and design of broiler houses can reduce poultry mortality. Also, this experiment obtained that RNG k-ε models are the most effective performers (error -6.2%) in the simulated ventilation case.

  Simulation of the ambient environment by CFD techniques can facilitate the research for thermal comfort. Experiment shows that indoor temperature and airflow can affect indoor occupations [23] [24].

### 2.3 CFD Applications for Manufacturing

CFD is also widely applied in the manufacturing industry, often reducing the need for extensive and expensive experiments [25]. Numerical analysis at an early stage can reduce prototype hardware revision and shorten the production cycle of products. CFD simulation in manufacturing can reduce the impact of harmful air on operators [26] [27] and ensure the successful production process [28].

- **Chip Manufacturing Factory**

  Due to the variety of raw materials and solvents used in a chip manufacturing factory and the production process's complexity, manufacturing factories often generate highly toxic gases,
which can be highly harmful to operators. The CFD technique simulates the velocity and concentration fields of the HCl gas stream and compares them with the validation data, allowing the factory to analyze the occupational hazards and control measures and obtain the best control measures [26].

- **Carbon Fiber Manufacturing Factory**

In carbon fiber manufacturing, it is significant to stabilize the ventilation of the precursor release gas for the production of carbon fiber. CFD can model a carbonization furnace for the polyacrylonitrile-based (PAN) carbon fiber manufacturing process, which guarantees that the carbonization homogeneity process runs successfully [28].

### 2.4 CFD Applications for Manufacturing Processes

CFD is widely used in manufacturing processes. CFD enables the modeling and analysis of four key features of DPRS (Dense particulate reaction system), namely high particle concentration, intense gas-particle/particle-particle interactions, complex chemical reactions, and significant mass and thermal transmission, according to the Eulerian-Eulerian and Eulerian-Lagrangian methods [29]. In the food production process, CFD simulates the drying, sterilization, mixing, and refrigeration steps in the production process, which allows for successful food production[30].

### 2.5 Advantages of CFD

CFD advantages are reduced error, energy savings, high simulation accuracy, and lower cost to physical prototyping.

- **Reduced Error**

Modern precision manufacturing processes often require production equipment with high positioning accuracy (e.g., CNC machines) to work in controlled environments with reasonable
airflow and a comfortable ambient temperature. Thermal errors in CNC machines usually come from uneven heating of individual components, resulting in component deformation leading to changes in the distance between the workpiece and the tool. According to statistics, 50% to 70% of the errors in CNC machine tools come from heat [31]. This experiment utilizes CFD technology to precisely simulate the surrounding environment of the CNC machine tool. This information can be used for the timely activation of the HVAC system to cool down the CNC machine tool and achieve the purpose of reducing machine errors.

Simultaneously, comfortable thermal conditions and airflow enable operators to stay motivated and reduce errors due to improper operator actions [23] [24]. Proper temperature and airflow can ensure that high-precision machines operate in a stable and optimal condition, thus improving the accuracy of the machines [31].

- **Lower Cost for Manufacturing**

In the manufacturing field, if one wants to obtain a specific flow field situation, there are generally three methods available: 1. obtaining the flow of a fluid by experimental methods (field observations); 2. pure mathematical models of fluid mechanics to describe the airflow; and 3. computer modeling simulation to describe fluid flow. Due to the high cost associated with the first method (e.g., wind tunnel tests) and the difficulty of solving the second method (fast airflow speed and the presence of large amounts of turbulence) [32], an increasing number of fields apply CFD technology to solve 3D fluid flow problems with complex turbulence, and an increasing number of fields apply CFD technology to solve 3D fluid flow problems with complex turbulence in the built environment [33].

- **Decreased Energy Consumption and High Simulation Accuracy for Manufacturing**
In the manufacturing industry, traditional HVAC systems are controlled by thermostats. Due to the distance of the thermostat from operators and CNC machines, the thermostat does not reflect the internal temperature change in a real manufacturing environment. CFD has unique advantages and well-established successes simulating airflow and temperature [34][35]. CFD simulates the ambient environment to accurately detect and monitor the thermal gradient and airflow around CNC machines, which provide the opportunity for energy savings for manufacturing, such as more proper timing of HVAC start-up and a reasonable layout of laboratory facilities [36].

2.6 Thermal Issues in Manufacturing

As CNC machine tools develop, modern CNC machines need to manufacture precision workpieces of different materials with different coefficients of thermal expansion. Because temperature affects cause up to 75% of the overall geometric errors of machined workpieces (other errors are kinematic error, thermomechanical error [37], dynamics, and motion control), it is concluded that one of the primary error sources in manufacturing is the constantly changing ambient temperature [38]. For machine tool development, temperature control remains a critical requirement for high precision manufacturing. Modern industrial production plants are all are controlled by HVAC systems; CFD is very widely applied in HVAC systems and has also achieved significant achievements [39]. So, the control of CNC machine tool error by CFD technology is one of the starting points of this experiment.

2.7 HVAC System and Thermal Comfort

The HVAC system operates on the principle of pulsated air [40]. By partially recycling the hot air from the room and sending cool air into the room to accomplish the temperature control in the room, the HVAC system becomes an essential system for the control of the indoor environment
in a building. However, HVAC systems have disadvantages such as noise, cold airflow, and vertical air temperature gradients, which can lead to discomfort, especially by female occupants in summer [41].

On this basis, the concept of thermal comfort is introduced. In an indoor environment, unacceptable thermal comfort conditions can cause reduced efficiency and an increased likelihood of operation errors [42]. The study of thermal comfort can help to detect and optimize HVAC systems. In a CFD simulation of a Korean university classroom, the influence of airflow and temperature on the average skin temperature of students was derived from the research, and it was found that students preferred fluctuating room temperature to constant room temperature [43].

The predicted mean vote (PMV), developed by Povl Ole Fanger at Kansas State University and the Technical University of Denmark, was adopted as an empirical fit to the ISO standard for human thermal comfort sensations [44]. The PMV is an index (shown in Figure 1) that CFD can simulate to predict people's hot and cold sensations in a room [45]. The values range from -3 to +3.

![PMV index](image)

**Influencing factors:**

- Air temperature
- Airflow
- Clothing Insulation
- Metabolic Rate
- Mean Radiant Temperature
- Relative Humidity

Figure 1: PMV index.
2.8 Digital Twin

In the 21st century of intelligent manufacturing and Industry 4.0, the digital twin is an important technology. The digital twin realizes the connection between the physical world and the virtual world of the network [46].

Currently, digital twin applications in the industry include product lifecycle management (PLM), from product involvement to final product operations. In the manufacturing industry, digital twins are utilized to improve employee safety [47], and replace current structural life-management approaches and the prediction of workpieces damage [48]. This experiment investigates the feasibility of experimenting with digital twins for real-time and remote inspection of environments by using CFD technology—the digital twin stores data in the cloud, allowing for unlimited data storage for the manufacturing industry.

The combination of CFD technology simulation accuracy and a digital twin real-time monitoring and management system can bring tremendous benefits to the manufacturing industry. For example, when the digital twin monitors a manufacturing plant, a system can detect a drastic change in the temperature of a CNC machine production line in a particular area. Factory managers know that the CNC machine in that area is in an undesirable condition and can immediately shut down the production line and repair the line, thus avoiding losses due to malfunction of CNC machines.

2.9 Research Goal and Objective

From the above literature (Table 1), it can be observed that the current research in the manufacturing area of CFD technology is limited to the production process (e.g., cement, food) and entire environment simulation. However, the application of CFD to evaluate the impact of high precision equipment from environmental changes has not been investigated. Alternatively,
the internal environment of high-precision equipment (e.g., EUV) has been investigated, but the environment around the equipment (outside the machine) has not been researched.

Table 1: The scope of the reviewed articles with CFD applications.

<table>
<thead>
<tr>
<th>Reference number</th>
<th>Scope</th>
<th>Airflow simulation</th>
<th>Temperature simulation</th>
<th>Related to manufacturing</th>
</tr>
</thead>
<tbody>
<tr>
<td>[15]</td>
<td>Isolation room</td>
<td>x</td>
<td></td>
<td></td>
</tr>
<tr>
<td>[16]</td>
<td>Shopping mall</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>[17]</td>
<td>Human body</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>[18]</td>
<td>Kyoto city</td>
<td></td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>[20]</td>
<td>Residential indoor</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>[21]</td>
<td>Public housing</td>
<td></td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>[22]</td>
<td>Office</td>
<td></td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>[23]</td>
<td>Electronic factory</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>[25]</td>
<td>Board</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>[26]</td>
<td>Clean room</td>
<td>x</td>
<td></td>
<td></td>
</tr>
<tr>
<td>[27]</td>
<td>Cement industry</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>[29]</td>
<td>Dense gas particulate</td>
<td></td>
<td>x</td>
<td></td>
</tr>
<tr>
<td></td>
<td>reaction system</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>[30]</td>
<td>Food processing industry</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>This experiment</td>
<td>Machining laboratory</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
</tbody>
</table>

Indoor environmental changes can cause many problems (high precision equipment structure alteration) in manufacturing, while CFD technology has advantages in simulating environmental accuracy. This experiment combines both above with a new approach: CFD technology simulates the machine laboratory environment to monitor the overall environmental variations of a laboratory (the S3 laboratory) and the environmental variations around high-precision equipment (CNC machine). Accurate results from CFD technology are used to achieve improved machine accuracy, operator thermal comfort, and energy savings. The ultimate goal of this research is to investigate the machining laboratory's environmental conditions to inform the design of precision manufacturing strategie.
Chapter 3: Methodology

3.1 Test Room and Flow Domain

The laboratory where the temperature and airflow for this experiment are investigated is the S3 laboratory (a machining laboratory). The machining laboratory is on the second floor of engineering building G (ENG), located at the University of South Florida (USF), Tampa, Florida, US. This research is conducted in a laboratory measuring 7.32 meters (24 feet) length, 4.27 meters (14 feet) width, and 3.29 meters (10.8 feet) height. The west wall of the S3 laboratory is the exterior wall, and the east wall is adjacent to another laboratory. This laboratory consists of a door (crevice present between the door and floor [gap inlet]), CNC machine, tables & shelves, washbasin, and HVAC system inlet and outlet vents. As shown Table 2.

Figure 2: SolidWorks model of the S3 laboratory. A and B: HVAC system inlet 1 and 2, respectively; C: light; D: table; E: HVAC system outlet; F: wash basin; G: operator; H: CNC machine; I: shelf.
This experiment is modeled in SolidWorks based on measurement lengths from the S3 laboratory, as shown in Figure 3. In order to improve the CFD simulation accuracy, this experiment adopts the approach of simplifying the complex geometric shape [49]. The complex human body and CNC machine shape are simplified into two rectangular shapes.

The research of the air domain for this experiment is the internal air domain of the laboratory. In the closed-door scenario, the air temperature and airflow speed in the hallway outside the laboratory is not considered since outside airflow is not in the experimental airflow domain, as shown in Figure 3. The temperature investigated in this experiment is the internal temperature of the laboratory, and the study area is focused on the region near the CNC machine (the working area). The outside laboratory temperature data is used as the boundary condition and is not simulated in the experiment. Even in the open door state, the air inside the laboratory is adjacent to the outside air.

Figure 3: The S3 laboratory simulated in SolidWorks and the air domain.
3.2 Experimental Facilities

This experiment uses the Govee thermometers (temperature is accurate to ±0.54°F/±0.3°C) to collect boundary and initial conditions and verify the accuracy of the CFD simulation as shown in Figure 4. An HTI thermal imager (273.85 K [0.7°C] thermal sensitivity) is also used. The measurement accuracy is ± 2 K [±2°C]); this device is used to capture temperature data for the human body, light and CNC machine. Additionally, a Pro Anemometer (airflow speed meter for measuring airflow speed range 0.30 m/s – 30 m/s) is used to acquire indoor airflow data to collect data for the boundary conditions and validate the CFD simulation.

Figure 4: A and B is the location of the Govee thermometer; point A is used to collect boundary conditions data; point B is used to verify the accuracy of the CFD simulation.
3.3 Overview of Methodology

An overview of the methods used for this experiment is shown in Figure 5. The research purpose of this experiment is to simulate the environment of the S3 laboratory using the CFD technique and then extend it to a larger scale manufacturing industry. Therefore, the main subject of this experiment is the air temperature distribution and airflow velocity in the experiment room.

First of all, the researcher enters the laboratory and conducts a preliminary analysis of the laboratory's internal environment (e.g., location of air inlets, heat-generating facilities in the room). Based on the analysis, the research involves a theoretical analysis of the air temperature distribution and airflow velocity. Then proceed to the analysis and establishment of the airflow model (Section 3.4). The pros and cons of different simulation models are discussed at the theoretical level to determine the required model. Then an attempt is made to optimize the disadvantages (Section 3.4.6). After the airflow step, this experiment discusses the thermal generation and thermal transfer theory and begins to prepare the required data collection (Section 3.5). After observing the laboratory's internal environment, the preparation of the laboratory data collection begins to be considered. Assumptions and optimizations are conducted based on the initial and boundary conditions required for the theoretical analysis. After listing the control experiments in different scenarios, the research started with SolidWorks Computer Aided Design (CAD) modeling of different control experiments (e.g., different cases of open and closed doors). CAD models are imported into Ansys fluent 2020 R1, and then a meshing step (Section 3.6) is performed to analyze the mesh accuracy (element quality, aspect ratio and independent study). The airflow and temperature models required for this experiment are optimized and selected from a practical level based on their applicability.
When the simulation is finished, this experiment compares the simulated data and the validation data (chapter 4). If the simulation result is not consistent with sensor data, this experiment can be performed with four methods to make the simulation results more accurate: replace another model or meshing method, polynomial interpolation to reduce discretization error (Eh) in CFD and balance method. The researcher also has to revisit whether the assumptions based on the theoretical analysis are reasonable. Novel numerical procedures were introduced in the Eh reduction research [50]. The study used the polynomial interpolation with the repeated Richardson extrapolation (RRE) calculation method to characterize the variables requiring improved accuracy into five given types. Then the RRE reduction Eh procedure was applied. The final results were examined by a linear problem and three nonlinear problems. The third approach is: since a higher stiffness coefficient can improve the accuracy of the simulation and a reduced efficient fluid time step can increase the resolution [51], a balance between computational accuracy and computational
time can be achieved by altering the efficient fluid time step (1 millisecond to 5 milliseconds) and stiffness coefficient values ($5 \times 10^4$ N/m to $5 \times 10^6$ N/m).

The next step is to investigate the effect on laboratory temperature and airflow in different cases to further explore its range of applicability. If the data are within a reasonable margin of error, the argument of this research is verified.

3.4 Airflow Investigation

3.4.1 HVAC System in the Machining Laboratory

There are two 100% outside air handling units (AHU) that serve all the laboratories in the ENG building, including ENG 229H (S3 laboratory) (personal communication, February 17, 2021). Both units supply air to the same pipe that distributes the air to all laboratories. Units are designed to deliver supply air at 285.93 K (55°F).

The air handling system adjusts the airflow and reheats the air for occupant comfort through Supply Air Terminals (SATs). The air in the laboratory is exhausted to the atmosphere through the fume hood exhaust fans (outlet). The amount of air exhausted from the air conditioning system is controlled by Exhaust Air Terminals (EATs). SAT in the S3 laboratory delivers a total of 17 m$^3$/min (700 CFM). EAT exhausts a total of 19.82 m$^3$/min (600 CFM) and 2.82 m$^3$/min (100 CFM) coming from the door opening. SAT adjusts the dampers and reheats the valves to maintain the laboratory temperature at the set point. The EAT tracks the airflow (in cubic meters per minute) provided by the SAT and adjusts the dampers to maintain a negative deviation of 2.82 m$^3$/min (100CFM).
3.4.2 Airflow in Machining Laboratory

The airflow velocity is always available and stable at the air inlets based on the actual measured airflow data. When the air conditioner turns to cold mode, the airspeed at the inlets becomes larger, and the temperature decrease.

This experiment divides the airflow situation into two cases:

- **Closed Door Condition**

  Two inlets and one outlet regulate the inside air domain of the laboratory as shown in Figure 6. At the same time, due to a gap between the door and the floor, the air pressure outside the laboratory is larger than the air pressure inside the laboratory, resulting in a tremendous gap inlet wind speed (4.3 m/s - 4.6 m/s) entering the laboratory.

- **Open Door Condition**

  Due to the door opening, the air pressure outside the laboratory still pushes the air into the laboratory, but the airflow speed of the door inlet drops to 0.3 m/s.

![Figure 6: SolidWorks model of the S3 laboratory. A and B: HVAC system inlet 1 and 2; C: outlet; D: gap inlet.](image)

3.4.3 Assumption

During the development of the model, the following assumptions are made:

1. The walls are adiabatic.
2. The walls are 0 thickness.

3. The air domain is incompressible airflow.

4. The outlet is the only way to exhaust the airflow out of the machining laboratory.

5. The turbulent viscosity is isotropic[52].

6. Steady turbulent flow has applied the airflow in the machining laboratory

7. The machining laboratory objects do not generate heat, except for the experimental research subjects (CNC machine, the operator, and the lights)

3.4.4 Airflow Model

Theoretically, according to driven forces (viscosity, inertial, etc.) of airflow, fluids are usually divided into laminar (viscosity forces are dominant) and turbulence models (inertial forces are dominant) [53]. Due to the low viscosity of air (18.5 μPa·s) [54], the turbulence model is adopted in this experiment. Turbulent flow is characterized by eddies, swirls, and flow instabilities [53].

The Reynolds number includes both static and kinetic properties of the fluid [55]. However, since this experiment is concerned with dynamic analysis, the Reynolds number is defined as a flow characteristic in this experiment. According to the Reynolds number equation (1), the ratio of inertial force to viscous force is the Reynolds number [56]. The numerator of Reynolds’ formula is the inertial force, and the denominator represents the viscous force. So, lower viscosity and higher airflow inertia increases the Reynolds number. In this experiment, the air viscosity is very small (water is 50 times more viscous than air) [57] [58].

The Reynolds number, \(N_{Re}\), is defined as shown in equation (1) [59]:

\[
N_{Re} = \frac{\rho v d}{\mu} = \frac{\text{inertial force}}{\text{viscous force}}
\]  

(1)

where \(\rho\) is density, \(v\) is velocity, \(d\) is diameter, and \(\mu\) is viscosity.
The Reynolds number has a significant effect on the velocity field, static pressure and turbulence characteristics [60]. In solving practical problems with turbulence models, the governing equation and numerical method are usually used to calculate turbulent viscosity [61]. One of three models are usually chosen: the Reynolds averaged Navier-Stokes equations (RANS), the large vorticity simulation equations (LES) or the Direct Numerical Solution (DNS) model. The primary purpose of turbulence modeling is to predict the mean fluid velocity, pressure, and temperature fields. Due to the turbulence modeling, the CFD calculation process does not need to be completed to calculate the turbulence modes as a function of time (LES and RANS) and does not need to calculate Navier-Stokes equations for every value of fluctuation (DNS).

- **DNS**

  DNS is appropriate for solving minor problems with simple geometric shapes. DNS directly attempts to solve the Navier-Stokes equation, which usually requires considerable computing capabilities [62]. Economically speaking, DNS is not suitable for this experiment.

- **LES**

  In the LES model, the smallest scale of eddies is removed from the calculation, and instead, the largest, most energy containing scales gets resolved [63]. Engineers consider the LES model in some engineering problems that anticipate unsteady heat transfer problems. The advantage of the LES model is that it can calculate the three-dimensional flow field with fewer meshes, so the computational power is less than that of DNS [64]. In industrial applications, the application of LES mode in CFD requires high performance computing because LES is several orders of magnitudes higher in terms of memory (RAM) and CPU time compared to RANS computing [65]. Therefore, this LES mode is not used in this experiment.

- **RANS**
The averaging operation of Navier-Stokes equations (both steady-state and dynamic flows) to obtain an average equation of fluid flow is called RANS [63]. In the situation where the full turbulence scale is modeled, the RANS model calculates the mean flow quantities, which results in a significant reduction in computational effort [63]. RANS model is applied in most industrial simulations because of its low computational cost and suitable accuracy compared to DNS and LES [66]. This model utilizes the Navier-Stokes equations and is based on the assumption that turbulence is isotropic [67]; RANS airflow does not expand the air due to contact with walls or get cut off by shear and swirling airflows. So, this experiment uses the RANS model (Reynolds-averaged Navier-Stokes equations).

3.4.5 Viscous Model

Under the RANS turbulence model family, there are two two-equation viscosity models. The viscous models considered for this research consisted of the k-epsilon (k-ε) model and k-omega (k-ω) model (as shown in equation [2-5]) [68].

The turbulence kinetic energy ($k$) in k-ε model is defined as shown in equation (2)[69]:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \tag{2}$$

Rate of dissipation ($\varepsilon$) is defined as shown in equation (3) [69] :

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + \frac{C_1 \varepsilon}{k} \left( G_k + C_3 \varepsilon G_b \right) - C_2 \rho \frac{\varepsilon^2}{k} + S_\varepsilon \tag{3}$$

$G_k$ is the generation of turbulence kinetic energy from average velocity gradients in turbulence kinetic energy. $G_b$ is the generation of turbulence kinetic energy from buoyancy in turbulence kinetic energy. $Y_M$ is the contribution of the fluctuating dilatation in turbulence kinetic energy. $C_{1\varepsilon}$, $C_{2\varepsilon}$, and $C_{3\varepsilon}$ are constants. $\sigma_\varepsilon$ and $\sigma_k$ are the turbulent Prandtl number of $\varepsilon$ and $k$. $S_\varepsilon$ and $S_k$ are user-defined source terms.

The turbulence kinetic energy ($k$) in k-ω model is defined as shown in equation (4) [70]:

[21]
\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \Gamma_k \frac{\partial k}{\partial x_j} \right] + G_k - Y_k + S_k \quad (4)
\]

The specific dissipation rate (\(\omega\)) is defined as shown in equation (5):

\[
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} (\rho \omega u_i) = \frac{\partial}{\partial x_j} \left[ \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + S_\omega \quad (5)
\]

\(G_\omega\) is the generation of \(\omega\). \(\Gamma_k\) and \(\Gamma_\omega\) is the effective diffusivity of \(k\) and \(\omega\). \(Y_k\) and \(Y_\omega\) is dissipation of \(k\) and \(\omega\) due to turbulence. \(S_k\) and \(S_\omega\) are user-defined source terms.

The k-\(\varepsilon\) model is commonly applied to simulate the average flow characteristics [71], while k-\(\omega\) is usually used to capture the effect of turbulent flow conditions [72]. Although the transport variable used in the two models is the turbulent kinetic energy (\(k\)) that determines the energy in the turbulent flow, different approaches are applied to the dissipation rate. The dissipation rate (\(\varepsilon\)) of the k-\(\varepsilon\) model determines the dissipation rate of turbulent kinetic energy. In contrast, the dissipation rate (\(\omega\)) determines the dissipation rate per unit of turbulent kinetic energy.

Compared with the k-\(\varepsilon\) model, the k-\(\omega\) model has good convergence and robustness [73]. This model has advantages in simulating objects with complex geometrical appearance and has a wide range of applications in the manufacturing industry [74]. The increase in turbulence is manifested as an effective fluid viscosity increase, and the Reynolds stress is proportional to the average velocity gradient through that viscosity [29]. In the case of airflow with adverse pressure gradient, flow field with strong curvature or jet flow, the simulation result of the k-\(\varepsilon\) model cannot be calculated accurately, but these cases are not considered in this experiment, so this disadvantage can be disregarded.

Compared with the k-\(\varepsilon\) model, the k-\(\omega\) model is easier to obtain results during the solution process. The k-\(\omega\) model simulates near-wall interactions more precisely than k-epsilon models. In the case of a low Reynolds coefficient, the k-\(\omega\) model has an advantage [74]. However, it is
challenging to converge in the k-ω model since the k-ω model is more nonlinear than the k-ε model [75]. The k-ω model presents the problem of over-predicting shear stresses for adverse pressure gradients; meanwhile, the k-ω model is more sensitive to initial/boundary conditions [76]. In a representative comparison of this experiment, it can be observed that the k-ε model converges better than the k-ω model (shown in Figure 7 [a] and [b]). Since the Reynolds coefficient of air is high (ranging from $3 \times 10^3$ to $4 \times 10^4$ [77]) and this experiment does not focus on the airflow near the wall, this experiment utilizes the k-ε model airflow model.

Figure 7: Convergence comparison of the (a) k-ε model and (b) k-ω model.
3.4.6 Optimization Solutions for k-ε Model

- **Realizable k-ε model**

In this experiment, a realizable model is selected for the airflow turbulence model to improve computational accuracy since the realizable k-ε model contains new formulation for the turbulent viscosity [78]. With the realizable k-ε model, ANSYS can precisely predict the spreading rate of planar and circular jets. Also, realizable k-ε yields accurate results in airflow involving rotation, boundary layer under strong adverse pressure gradients, separation, and recirculation [79]. The realizable k-ε model gives better results when considering simulating the turbulent eddy viscosity and the transport equation for the kinetic energy (k) and dissipation rate (ε) since this equation (equation [6-9]) is derived from the exact transport equation for the mean square vorticity fluctuations [80].

The realizable equation for improving the accuracy of ε is defined as shown in equation (6-9) [69]:

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j} (\rho k u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_m + S_k \tag{6}
\]

\[
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j} (\rho \varepsilon u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S_\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\varepsilon \nu}} + C_1 \frac{\varepsilon}{k} C_2 G_b + S_\varepsilon \tag{7}
\]

\[
C_1 = \max \left[ 0.43, \frac{\eta}{\eta + 5} \right] \tag{8}
\]

\[
\eta = S_k \frac{k}{\varepsilon} \tag{9}
\]

Y_{m} is the contribution of fluctuating expansion in compressible turbulent flows to the total dissipation rate.

- **Enhanced wall treatment**

Enhanced wall treatment is a near-wall modeling approach that combines two modeling approaches (two-layer modeling and enhanced features) [81]. Specifically, in the k-ε model, the
enhanced wall treatment is combined with different wall functions for solving (this is why it is called a two-layer method). The enhanced wall treatment applies one equation relations to evaluate the laminar sublayer of the fine mesh and provide a way to extend the scalability of the data compared with the standard avoidance function (via Y+ and enhanced wall treatment) [82]. The near-wall meshing must be reasonable; otherwise, it will increase the computational effort of the computer.

3.5 Heat Generation in the Machining Laboratory

3.5.1 Heat Generation

Since the experiment investigates the internal environment of the laboratory, this experiment assumes that the heat inside the laboratory is generated by the CNC machine, the operator, and the electric lights.

- **CNC Machine**

The CNC machine model in the laboratory is the Levil LMV-400 with an all-aluminum alloy structure (Dimensions: W 0.84 meters [33 inches] * H 1.72 meters [68 inches] * D 0.94 meters [37 inches]) as shown in Figure 8 (a).

Figure 8: (a) CNC machine and (b) heat generated inside of CNC (c) heat generated by front side of CNC machine; (d) heat generated by the back side of CNC machine.

- **Heat generated from inside the CNC machine**
The heat generation of the CNC machine mainly comes from the contact between the cutting tool and the material, and the rotation of the spindle shown in Figure 8 (b). The faster the rotation speed of the spindle, the more power is required and the higher the heat generation.

- **Heat generated from outside the CNC machine**

The heat generated by the CNC machine and transferred (thermal convection) to the laboratory is shown in Figure 8 (c) and Figure 8 (d). There is an open area at the back of the CNC machine. The heat from the inside of the CNC machine can be transferred to the outside through a medium (e.g., air). Even though the temperature between the material and the cutting tool is 376.05 K (102.9 °C) (Figure 8 [b]) from HTI thermal imager, this portion of the heat does not eventually enter the laboratory and raise the temperature in the laboratory. So, the heat generated by the CNC machine used in this experiment is the average of the various temperature points measured at the exterior (frame) temperature of the CNC machine.

- **Heat generated from Operator and Lights**

To conveniently display the temperature changes around the operator, a rectangular body is used in this experiment instead of an actual operator model. This experiment simulates the heat production of an adult male operator whose measurement temperature is 299.47 K (26.32 °C) as shown in Figure 9 (a). There are four sets of bulbs on top of the laboratory. Each bulb is 277V, 32W, and 3500 kelvin. The generated temperature (294.85 K [21.7°C]) is shown in Figure 9 (b).

3.5.2 Energy Equation

The heat transfer process from one area of material heat energy flowing to another region because of temperature differences is called heat transfer, resulting in temperature distribution and changes [83]. As materials warm up or cool down, the heat transfer changes the kinetic energy. The energy equation (equation [10-12]) is needed in the process of heat transfer.
Figure 9: (a) Heat generation by the operator (b) heat generation by the operator light.

The Ansys fluent energy equation is modelled based on the solution to the following equations (10-12) [84]:

\[
\frac{\partial}{\partial t} (\rho E) + \nabla \cdot (\bar{v}(\rho E + p)) = \nabla \cdot [k_{\text{eff}} \nabla T - \sum_j h_j \bar{J}_j + (\tau_{\text{eff}} \cdot \bar{v})] + S_h \tag{10}
\]

where \( k_{\text{eff}} \) is the effective conductivity; \( \bar{J}_j \) is the diffusion flux of species \( j \); \( S_h \) is heat of chemical reaction and any other volumetric heat sources; \( k_{\text{eff}} \nabla T \) is energy transfer from conduction; \( \sum_j h_j \bar{J}_j \) is energy transfer from species diffusion; and \( \tau_{\text{eff}} \cdot \bar{v} \) is energy transfer from viscous dissipation. \( S_h \) includes chemical reaction heat, and any other volumetric heat source defined by the user.

In equation 10,

\[
E = h - \frac{p}{\rho} + \frac{v^2}{2} \tag{11}
\]

where sensible enthalpy \( h \) for incompossible airflow shown as equation 12:
where $Y_j$ is the mass fraction of species and $h_j$ shown as equation 13:

$$h_j = \int_{T_{ref}}^{T} c_{p_j}dT$$ (13)

where the numerical values $T_{ref}$ are determined by the solver and model applied by the user.

3.5.3 Heat Transmission Method

The heat-producing facilities of this experiment, such as lights, can exchange heat with the nearby cooler air (heat transfer by movement of a fluid [85]), while the air can take away the heat generated by the lights (move away from the source of heat). This heart transmission is thermal convection. The lights also release the acceleration of charged particles, and those particles transfer heat through electromagnetic radiation.

So, the heat generated in this experiment is both thermal radiation and thermal convection, which is mixed thermal conditions in ANSYS fluent. Table 2 shows the parameters required for mixed thermal conditions.

<table>
<thead>
<tr>
<th>Parameter of mixed thermal conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>• Heat transfer coefficient ($w/m^2\cdot k$)</td>
</tr>
<tr>
<td>• Free stream temperature ($^\circ C$)</td>
</tr>
<tr>
<td>• External emissivity</td>
</tr>
<tr>
<td>• External radiation temperature ($^\circ C$)</td>
</tr>
<tr>
<td>• Heat generation rate ($w/m^3$)</td>
</tr>
</tbody>
</table>

The heat generation considered in this experiment consists of thermal radiation (Equation 14) and thermal convection (Equation 15). The radiation equation for heat transfer (see Equation 14, where heat is transferred from object $i$ to object $j$ by radiation) is:

$$Q_{ij} = A_i \varepsilon_{hi} F_{ij} \sigma (T_i^4 - T_j^4)$$ (14)
where $\varepsilon_h$ is emissivity in heat radiation, $F_{ij}$ is the view factor in heat radiation, $T$ is the temperature in absolute units in heat radiation, $\sigma$ is the Stefan-Boltzmann constant in heat radiation, $\sigma = 5.67 \times 10^{-8} \text{ W/m}^2 \cdot \text{K}^4$, and $\varepsilon_{hl}$ is the emissivity (the ratio of the energy radiated from a material’s surface over the energy radiated from a black body’s surface). Thermal convection was also considered in the model with pertinent parameters summarized in Table 2.

The heat transfer equation coefficient (Table 2) is defined as shown in equation (15):

$$h_{\text{transfer}} = \frac{q}{\Delta T} \quad (15)$$

$h_{\text{transfer}}$ is heat transfer coefficient, $q$ is the heat flux in heat transfer and $\Delta T$ is the difference in the two contact surface temperatures. Additionally, the heat generation rate for thermal convection (equation 16) can be found as follows and is represented in units of (W/m$^3$):

$$\text{Heat generation rate} = \frac{\text{Power}}{\text{Volume}} \quad (16)$$

3.5.4 Linear Thermal Expansion

The components of CNC machine tools are sensitive to temperature changes, which affects the mechanical structure (frame) of the CNC machine [86]. Such structural changes lead to the appearance of thermal errors. According to prior research, thermal errors account for 75% of overall geometrical errors of machined workpieces [87]. Based on the change of the CNC machine frame, linear thermal expansion is applied in this experiment to calculate the mechanical structure deformation. This experiment considers the width and height of the CNC machine with alternative materials consisting of aluminum, iron, steel and stainless (Table 3).

Linear Thermal Expansion is defined as shown in equation (17) [88]:

$$dl = L_0 \alpha (t_1 - t_0) \quad (17)$$
where \( dl \) is the change in variation of length (m); \( L_0 \) is the initial length (m); \( \alpha \) is the linear expansion coefficient; \( t_0 \) is the initial temperature (°C or K) and \( t_1 \) is the final temperature (°C or K).

Table 3: Linear expansion coefficient of different materials, at normal temperature (20 °C).

<table>
<thead>
<tr>
<th>Linear expansion coefficient (°C(^{-1}))</th>
<th>Aluminum</th>
<th>Iron</th>
<th>Steel (hardened)</th>
<th>Stainless</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000023</td>
<td>0.000011</td>
<td>0.0000124</td>
<td>0.0000104</td>
<td>[89]</td>
<td></td>
</tr>
</tbody>
</table>

3.6 Meshing Method

Since the meshing method is the same, we use the simulation of 5/13/2021 as an example to introduce the meshing method for this experiment. To demonstrate the geometric object resolution and to ensure the accuracy of the high gradient region, this experiment uses the size control method [90]. The ANSYS Fluent meshing method is an intuitive way to apply the element size to any given face or group of faces, ensuring a consistent, high-quality mesh size for the region. This experiment uses an element size of 0.03 m and soft behavior for the 149 interior faces of the S3 laboratory as shown in Figure 10 (a). Element size of 0.06 m and the same soft behavior option are applied for the 9 exterior walls of the lab as shown in Figure 10 (b). The soft behavior option was chosen because the size is global and does not have the same strict size control as the hard behavior. Hard behavior causes the transition between the hard edge and face mesh to be abrupt [91].
Figure 10: (a) Face sizing method for interior faces of machining laboratory (b) face sizing method for exterior walls of machining laboratory.

By increasing the quality smoothing (by adjusting smoothing option) to the highest level, ANSYS Fluent statistics indicate that nodes are 425,483 and elements are 2,266,996. In the “Setup” step, the experiment applies the polyhedral method. Tetrahedral or hybrid mesh's numerical properties are not optimal. However, the polyhedral method takes advantage of the fact that the
polyhedral are surrounded by many neighboring points (Figure 11[a] vs Figure 11[b]), making the gradient approximation much better than the tetrahedral or hybrid mesh [92].

A high-quality mesh allows the computer to balance computational cost and refinement of simulation, so quality checking of the mesh is necessary [93]. This experiment is analyzed from three aspects: element quality, aspect ratio and independent study.

3.6.1 Element Quality

The element quality is a composite quality (range from 0 to 1), as shown in equation 17 [94]:

\[
\text{Element Quality} = C \left[ \frac{\text{Volume}}{\sqrt{\sum (\text{Edge Length})^2}} \right]^3 (17)
\]

where the element determines the value of C.

Element quality is the ratio of the volume to the square root of the cube of the sum of the square of the edge lengths. If it is 1, it is a perfect square. From the ANSYS Fluent numerical results, most of the element quality of this experiment is close to 1 as shown in Figure 12 (a). From Appendix A (ANSYS Fluent meshing report), the maximum value of element quality is 0.99, the minimum value is 0.21, the average value is 0.837, and the standard deviation is 0.095. From the ANSYS Fluent graphical results, the large blue area (close to 1) also indicates that this experiment has a fine meshing as shown in Figure 12 (b).
Figure 11: (a) Tetrahedral meshing method (b) polyhedral meshing method.
3.6.2 Aspect Ratio

Aspect ratio is the deviation of the component when the length of each side is equal, as shown in equation 18 [95]:

\[
\text{Aspect Ratio} = \frac{\text{Long Edge Length}}{\text{Short Edge Length}}
\]

As in equation 17, the aspect ratio value must be about 1. If the ratio is 1, the shape is a perfect square, and the shape will be distorted if the ratio becomes larger. So about close to 1, it
means the better quality of meshing [96]. From the ANSYS numerical results, most of the aspect ratios of this experiment are below 2.5 and close to 1 as shown in Figure 13 (a). From Appendix B, the minimum value of the aspect ratio of this simulation is 1.16, the maximum value is 10.93, the average value is 1.85, and the standard deviation is 0.46. From the ANSYS Fluent graphical results, the large blue area also demonstrates that this experiment is well-meshed as shown in Figure 13 (b).

Figure 13: (a) ANSYS Fluent numerical results of aspect ratio (b) ANSYS Fluent graphical results of aspect ratio.
3.6.3 Independent Study

This experiment utilized two approaches to complete the mesh independent study. First, the correlation between face sizing and the average and maximum airflow velocity of the outlet. Second, the relationship between face sizing and CNC machine temperature is identified.

As shown in Figure 14, this experiment reveals that as the face sizing decreases, neither the average airflow speed nor the maximum airflow speed changes drastically, indicating that the mesh processing of this experiment is accomplished reasonably. In the range of computational capacity, this experiment chooses the minimum face sizing (0.03 m) to make the simulation result accurate. As shown in Figure 15, the CNC machine temperature also does not change dramatically. It can also show that the mesh processing of this experiment is perfect.

![Independent study for outlet speed](image)

Figure 14: Independent study for average and maximum outlet airflow speed.
3.7 Modeling Ambient Conditions via CFD

3.7.1 CFD Simulation Scenarios

In this experiment, three control experiments (Table 4) were conducted to investigate the effect of different experimental results. Scenario 1 simulates the impact of operators on the factory environment during the operation of CNC machines in a manufacturing factory. Scenario 2 is to simulate the environmental changes in the factory when the manufacturing plant is shut down (CNC machine turns off at night and the operator leaves) and when the factory starts working (CNC machine starts during the day and the operator starts working). Scenario 3 simulates the impact of operators on the laboratory environment under open door conditions. In this case, the factory environment is influenced by both natural ventilation and the HVAC system. The three cases are also compared vertically, and the experimental results allow exploring the factory changes in different seasons and different air-conditioning temperature cases. The CFD models were developed in ANSYS Fluent 2020 R1 for three scenarios: (as shown in Table 4)
Table 4: Three scenarios

<table>
<thead>
<tr>
<th>Scenarios</th>
<th>Case</th>
<th>Door</th>
<th>CNC machine</th>
<th>Air flow</th>
<th>Heat Generation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Scenario 1</strong> (April)</td>
<td>Case 1</td>
<td>Door Closed</td>
<td>Machine ON</td>
<td>HVAC inlets and Gap inlet</td>
<td>Lights, CNC machine</td>
</tr>
<tr>
<td></td>
<td>Case 2</td>
<td></td>
<td>Machine ON</td>
<td>HVAC inlets and Gap inlet</td>
<td>Lights, CNC machine, Operator</td>
</tr>
<tr>
<td><strong>Scenario 2</strong> (May)</td>
<td>Case 3</td>
<td>Door Open</td>
<td>Machine OFF</td>
<td>HVAC inlets and Gap inlet</td>
<td>Lights</td>
</tr>
<tr>
<td></td>
<td>Case 4</td>
<td></td>
<td>Machine ON</td>
<td>HVAC inlets and Gap inlet</td>
<td>Lights, CNC machine, Operator</td>
</tr>
<tr>
<td><strong>Scenario 3</strong> (May)</td>
<td>Case 5</td>
<td>Door Open</td>
<td>Machine ON</td>
<td>HVAC inlets and Door inlet</td>
<td>Lights, CNC machine</td>
</tr>
<tr>
<td></td>
<td>Case 6</td>
<td></td>
<td>Machine ON</td>
<td>HVAC inlets and Door inlet</td>
<td>Lights, CNC machine, Operator</td>
</tr>
</tbody>
</table>

3.7.2 Scenarios Data

The data collection for the three Scenarios is Table 5, Table 6 and Table 7, which are used for the initial condition parameters as described in the previous section. The parameter measured with the temperature sensor is not measured at a time measurement, but rather the average of 30 minutes of stable data. The airspeed meter data is the average of multiple measurements at that point, respectively.

The measurements for case 1 and case 2 were obtained on April 16 2021. Case 3 and case 4 were collected in May 13 2021, and case 5 and case 6 data were gathered in May 10 2021.

3.7.3 Validation

The simulation results of the experiment are compared with the sensor data to obtain the accuracy of the simulation. This experiment uses the percentage error (equation 17) to verify the air temperature difference between simulation and sensor data.

Percentage Error Formula is defined as shown in equation (19) [97]:

\[
\text{Percentage Error} = \frac{|E - T|}{|T|} \quad (19)
\]

where \(E\) is the experimental (measured) value; and \(T\) is the theoretical (simulation) value.
Table 5: Case 1 parameter of airflow speed and temperature to case 6 parameter of airflow speed and temperature.

<table>
<thead>
<tr>
<th>Case</th>
<th>Inlet 1</th>
<th>Inlet 2</th>
<th>Gap inlet</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Airflow speed (m/s)</td>
<td>Temperature (K)</td>
<td>Airflow speed (m/s)</td>
</tr>
<tr>
<td>Case 1</td>
<td>3.2</td>
<td>292.37</td>
<td>2.6</td>
</tr>
<tr>
<td>Case 2</td>
<td>3.4</td>
<td>292.35</td>
<td>2.6</td>
</tr>
<tr>
<td>Case 3</td>
<td>3.5</td>
<td>293.77</td>
<td>2.4</td>
</tr>
<tr>
<td>Case 4</td>
<td>3.5</td>
<td>293.64</td>
<td>2.4</td>
</tr>
<tr>
<td>Case 5</td>
<td>3.5</td>
<td>294.42</td>
<td>2.4</td>
</tr>
<tr>
<td>Case 6</td>
<td>3.2</td>
<td>294.58</td>
<td>2</td>
</tr>
</tbody>
</table>

Table 6: Case 1 parameter of lights, CNC machine and operator to case 6 parameter of lights, CNC machine and operator.

<table>
<thead>
<tr>
<th>Heat transfer coefficient (W/m²·K)</th>
<th>Lights</th>
<th>CNC machine</th>
<th>Operator</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1 - 6</td>
<td>6.3</td>
<td>25</td>
<td>4.7</td>
<td>[98] [99] [100]</td>
</tr>
<tr>
<td>External emissivity</td>
<td>0.8</td>
<td>0.8</td>
<td>0.8</td>
<td>[101]</td>
</tr>
<tr>
<td>External radiation temperature (K)</td>
<td>294.77</td>
<td>296.65</td>
<td>299.47</td>
<td></td>
</tr>
<tr>
<td>Heat generation rate (W/m³)</td>
<td>496</td>
<td>1207</td>
<td>266</td>
<td></td>
</tr>
</tbody>
</table>
Table 7: Free stream temperature for the lights, CNC machine, and operator.

<table>
<thead>
<tr>
<th></th>
<th>Case 1</th>
<th>Case 2</th>
<th>Case 3</th>
<th>Case 4</th>
<th>Case 5</th>
<th>Case 6</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lights</td>
<td>293.09</td>
<td>293.09</td>
<td>295.74</td>
<td>295.58</td>
<td>296.5</td>
<td>296.9</td>
</tr>
<tr>
<td>CNC Machine</td>
<td>293.09</td>
<td>293.09</td>
<td>295.74</td>
<td>295.58</td>
<td>296.5</td>
<td>296.9</td>
</tr>
<tr>
<td>Operator</td>
<td>293.09</td>
<td>293.09</td>
<td>295.74</td>
<td>295.58</td>
<td>296.5</td>
<td>296.9</td>
</tr>
</tbody>
</table>
Chapter 4: CFD Results and Validation

4.1 Scenario 1: Case 1 and Case 2 CFD Results Comparison and Validation

Scenario 1 shows the variation in the temperature and airflow data throughout the experiment in a comparison of case 1 and case 2 from April 16, 2021 (temperature of S3 laboratory is 293.09K). This comparison experiment is designed to investigate the impact of operator access on laboratory temperature and air flow (case 1 without operator vs case 2 with operator).

4.1.1 Temperature Specification

Data from four points accomplish the temperature collection for this experiment. The location of the four points in the laboratory is represented in Appendix C. After the data became stable, each data point is derived from the average over half an hour of data. In case 1, point A is directly below the HVAC system inlet 1; point B is directly below inlet 2; point C is near the operator’s position, and point D is the temperature of the gap inlet. In case 2, point A, B and C are the same as case 1; point D is near the CNC machine.

The temperature contour (Figure 16 [a]) indicates that the heat in the laboratory comes from the lights and the CNC machine in case 1. Compared to case 1, case 2 added the operator's heat generation status, as shown in Figure 17 (a). Since the outside air pressure is greater than the inside air pressure of the laboratory, the airflow from the gap inlet into the laboratory is fast. At the same time, due to the high temperature outside, the airflow entering the laboratory from the gap inlet carries a large amount of heat as shown in Figure 16 (a) and Figure 17 (a). Due to the high velocity and high temperature of this gap inlet, this heat covered a large area (light blue area) of the laboratory as shown in Figure 17 (b). At the same time, since there is no heat-producing
equipment in the left area (there is a CNC machine producing heat on the right area), the temperature on the left side is lower and appears blue Figure 16 (b). The simulation results of the volume rendering and contour rendering are consistent.

Figure 16: (a) Temperature contour and (b) temperature volume rendering for case 1.
As shown in Figure 18 (a - d), the temperature varies from 292.3K to 298.3K in case 1 and 292.3K to 298.5K in case 2 throughout the laboratory. The contour plots of the temperature and airflow velocity are relatively uniform in the 2D plane. The lower temperature enters the room from the HVAC system inlets and accomplishes the temperature adjustments in the S3 laboratory. Since this experiment simplifies the human body into a rectangular body, the rectangular body in
Figure 19 (a) generates no heat while the rectangular body in Figure 19 (b) generates heat, indicating that the 2D plane figures are consistent with the 3D figure above (Figure 17).

The validity of the numerical model used in the CFD simulations was confirmed by the comparison of the experimental and validation data. Since case 1 point C (Figure 18 [a]) and case 2 point C (Figure 18 [b]) are at identical locations and located close to the CNC machine, this location can reflect the temperature variation in the working area of this experiment. From the variation in temperature (Figure 20 [a] point C vs Figure 20 [c] point C), it can be concluded that due to the operator’s entry into the room, the temperature around the CNC machine has an impact, which is raising the surrounding temperature by 0.6 K (293.14 K vs. 293.74 K). The simulated data at the same location increased by 0.1 K (293.52 K vs. 293.62 K), showing the same upward trend as the validation data.

Figure 18: Case 1 (a) temperature (292.3 K to 298.5 K) and (b) airflow speed (0 m/s to 4.27 m/s); case 2 (c) temperature (292.3 K to 298.5 K) and (d) airflow speed (0 m/s to 4.27 m/s).
Figure 19: 2D vertical view of temperature contour near CNC machine (a) without the operator for case 1 and (b) with the operator for case 2.

As mentioned in the methodology, this experiment explores the variation of the X (width deformation of CNC machine frame) and Z axes (height deformation of CNC machine frame) of the internal table of the CNC machine. Since the temperature increases by 0.6 K, the width (0.84 m) and height (0.97 m) of different materials will change differently due to the different coefficients of thermal expansion. The different variations are shown in Table 8 below. The variation in temperature causes the outward expansion of the CNC machine frame.

Table 8: Scenario 1 CNC machine deformation under different materials.

<table>
<thead>
<tr>
<th>Width deformation (mm)</th>
<th>Aluminum</th>
<th>Iron</th>
<th>Steel (hardened)</th>
<th>Stainless Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Height deformation (mm)</td>
<td>0.01159</td>
<td>0.00554</td>
<td>0.00625</td>
<td>0.005242</td>
</tr>
<tr>
<td>0.01338</td>
<td>0.00640</td>
<td>0.00721</td>
<td>0.00605</td>
<td></td>
</tr>
</tbody>
</table>
Figure 20: (a) Temperature and (b) airflow speed for case 1; and (c) temperature and (d) airflow speed for case 2.

<table>
<thead>
<tr>
<th></th>
<th>Simulation data(K)</th>
<th>Validation data(K)</th>
<th>Percentage error(%)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Case 1</strong></td>
<td><strong>Case 2</strong></td>
<td><strong>Case 1</strong></td>
<td><strong>Case 2</strong></td>
</tr>
<tr>
<td><strong>A</strong></td>
<td>293.85</td>
<td>293.52</td>
<td>293.50</td>
</tr>
<tr>
<td><strong>B</strong></td>
<td>293.52</td>
<td>293.62</td>
<td>294.10</td>
</tr>
<tr>
<td><strong>C</strong></td>
<td>293.52</td>
<td>293.76</td>
<td>293.14</td>
</tr>
<tr>
<td><strong>D</strong></td>
<td>295.24</td>
<td>293.76</td>
<td>295.24</td>
</tr>
</tbody>
</table>

In addition, the accuracy of the results was analyzed by a sensitivity study (the variable for the comparison experiment of scenario 1 is operator access.) of airflow velocity and airflow model to temperature variations at different points in the S3 laboratory. The simulated temperature in the laboratory was very similar to the measured temperature data (Figure 20 [a] and Figure 20[c]); the
average percentage error amounted to 0.11% (case 1) and 0.08% (case 2) and the respective percentage errors of each point is shown in Table 9. Overall, the error is in a range that satisfies engineering requirements [102].

4.1.2 Airflow Specification

As shown in Figure 21 (a) and Figure 22 (a), the streamlines of case 1 and case 2 represent the air distribution only from the gap inlet into the S3 laboratory. As described above, the gap inlet airflow velocity is high, which can blow the air to the north wall and then diffuse it, thus affecting the airflow distribution in the internal area of the laboratory (near the north wall area). The HVAC system brings the cold air into the laboratory through two air inlets. Finally, the mixed airflow of the laboratory is sent out by the outlet as shown in Figure 21 (b) and Figure 22 (b). The large light pink area of case 1 and 2 show areas of relatively stable airflow. In terms of operator perception, the simulated airflow distribution matches the observed laboratory conditions. Due to the relatively low width of the operator, there is no significant difference in the airflow distribution when comparing case 1 and case 2.

The airflow velocity changes from 0 m/s to 4.27 m/s in case 1 and 0 m/s to 4.27 m/s in case 2 throughout the laboratory as shown in Figure 18 (b) and Figure 18 (d). The simulated airflow speed showed deviations from the measured data as shown in Figure 20 (b) and Figure 20 (d). The error range of air flow velocity in case 1 is from 0 m/s to 0.65 m/s. The error range of the laboratory air flow speed in case 2 is from 0.08 m/s to 0.34 m/s, as shown in Table 10.
Table 10: Airflow speed error of case 1 and case 2.

<table>
<thead>
<tr>
<th></th>
<th>Simulation Data (m/s)</th>
<th>Validation Data (m/s)</th>
<th>Error (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Case 1</td>
<td>Case 2</td>
<td>Case 1</td>
</tr>
<tr>
<td>A</td>
<td>2.70</td>
<td>2.74</td>
<td>2.50</td>
</tr>
<tr>
<td>B</td>
<td>0.19</td>
<td>0.20</td>
<td>OOR</td>
</tr>
<tr>
<td>C</td>
<td>0.74</td>
<td>0.08</td>
<td>0.70</td>
</tr>
<tr>
<td>D</td>
<td>2.25</td>
<td>0.58</td>
<td>1.60</td>
</tr>
<tr>
<td>E</td>
<td>0.60</td>
<td>0.71</td>
<td>0.60</td>
</tr>
<tr>
<td>F</td>
<td>0.66</td>
<td>NA</td>
<td>0.80</td>
</tr>
</tbody>
</table>

Note: OOR represents out of range.

Figure 21: (a) Airflow streamline only from gap inlet and (b) airflow volume rendering for case 1.
Figure 22: (a) Airflow streamline only from gap inlet and (b) airflow streamline in volume rendering for case 2.
The airflow speed at point B in case 1 and case 2 is the airflow speed at the top of the CNC machine. Since the airflow meter for measuring airflow speed ranges from 0.30 m/s to 30 m/s (0.67~67.1 mph) [103], the sensor data cannot be utilized at point B. Therefore, the difference between the simulated airflow speed and the validation speed is neglected. When the airflow is obstructed by some objects (such as chairs), the air turbulence is reflected at a reduced speed, which can be the source of error at point C for case 2. Extraneous objects in the laboratory are not taken into account in the modeling as omitting extraneous objects can save simulation time. Points E and F in case 1, and points D and E in case 2 are affected by the turbulence formed near the wall, which affects their accuracy. The best model for turbulence near the wall is the k-ω model [104], but the k-ε model is a better choice for this experiment because it is more concerned with the overall airflow in the S3 laboratory and offers better convergence. The k-ε model’s advantage is the simulation of point A in case 1 and in case 2, where the airflow simulation is very successful because point A is not disturbed by any other objects.

4.2 Scenario 2: Case 3 and Case 4 CFD Results Comparison and Validation

Scenario 2 shows the variation in the temperature and airflow data throughout the experiment in a comparison of case 3 and case 4 from May 13 2021 (temperature of S3 laboratory is 295.15K). The research topic for the scenario 2 experiment is to compare the laboratory operator who leaves and shuts down the CNC machine with the situation where the person operates the CNC machine. The thermal generation variation is simulated by both the operator and the CNC machine. Also, this experiment can be compared vertically with scenario 1. The comparison of these two experiments (scenario 1 vs. scenario 2) are designed to investigate the influence of the CNC machine at different times (seasons) on the air temperature and airflow speed in the laboratory.
4.2.1 Temperature Specification

As the season’s change, the parameters of the air conditioning system in the laboratory change. The set temperature of the laboratory thermostat rises (between April and May), bringing alterations to the temperature throughout the laboratory. In this condition, the accuracy of this experiment needs to be supported by more data. This provides the target for scenario 2 of control experiments. This comparison experiment uses four temperature test points to verify the accuracy of the experiment. As in the previous scenario, the data are averaged over half an hour after temperature stabilization at each point. The locations of the four points are shown in appendix D.

Comparing Figure 23 (a) and Figure 24 (a), it can be seen that this simulation experiment reasonably simulates the difference in scenario 2, that is, the introduction of two more sources of thermal generation in case 4 - the operator and the CNC machine. Although the scenario 2 air temperature outside the laboratory is higher than scenario 1 (seasonal change) and the airflow velocity remains the same, the bottom view (Figure 23 (b) and Figure 24 (b)) of the laboratory indicates that the gap inlet can still blow the outside air to the north wall. The airflow of the gap inlet affects the air temperature distribution near the north wall area. In case 3 and case 4, the air temperature distribution is different to the left and right of the CNC machine due to the obstruction of the CNC machine (light blue area for the low temperature on the left and light green area for the high temperature on the right) as show in Figure 23 (b). Such a temperature change is confirmed in the next step of data validation. The bottom view of case 4 also shows that the heat generation by the CNC machine and the operator is included in case 4.
Figure 23: (a) Temperature contour and (b) bottom view of temperature volume rendering in case 3.
Figure 24: (a) Temperature contour and (b) bottom view of temperature volume rendering in case 4.

Figure 25 (a) and Figure 25 (c) show the air temperature variation in the 2D dimension vertically for case 3 and case 4. Case 3 temperature variation is from 293.4K to 299.7K with an average percentage error of 0.15% (Table 11); Case 4 temperature variation is from 293.4K to
299.9K with an average percentage error of 0.11%; The percentage errors for each of the four points are listed separately (see Table 11). In the vertical 2D plane views (Figure 26 [a] and [b]) of the CNC machine, it can be seen that when the operator enters and the CNC machine starts, the environment around the CNC machine is changed, which is consistent with the 3D figures above (Figure 23 and Figure 24). There is reasonable consistency between CFD predictions and experimental results. As shown in Table 11, the error range of case 3 is slightly larger than the error range of case 4.

Figure 25: (a) Temperature (293.4k to 299.7k) and (b) airflow speed (0 m/s to 4.24 m/s) for case 3; (c) Temperature (293.4k to 299.9k) and (d) airflow speed (0 m/s to 4.3 m/s).
Table 11: Temperature percentage error for case 3 and case 4.

<table>
<thead>
<tr>
<th>Simulation data(K)</th>
<th>Validation data(K)</th>
<th>Percentage error(%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Case 3</td>
<td>Case 4</td>
</tr>
<tr>
<td>A</td>
<td>295.50</td>
<td>295.93</td>
</tr>
<tr>
<td>B</td>
<td>295.08</td>
<td>295.33</td>
</tr>
<tr>
<td>C</td>
<td>295.23</td>
<td>295.97</td>
</tr>
<tr>
<td>D</td>
<td>295.26</td>
<td>295.62</td>
</tr>
</tbody>
</table>

When the operator enters the laboratory, the temperature near the CNC machine drops by 0.17 K (Figure 27 (a) point C vs Figure 27 (c) point C). By comparing case 2 and case 4 vertically, the experimental results of scenario 2 are opposite to the results of scenario 1 when the operator enters the S3 laboratory and turns on the CNC machine.

The main reason is that the laboratory HVAC system is switched on for cooling. From Figure 28 and the airflow meter, when the HVAC turns to cold mode, the airspeed at the inlets becomes larger, and the temperature decreases. The boundary conditions (temperature) variation at the air inlets of the air conditioning system is shown in Figure 28. The data collection time is 30 minutes, from the start time of case 4 to the end time of case 4 (one sample collected per minute). The measured temperatures of inlet 1 and inlet 2 are over the same period. In order to eliminate the effect of people just entering the laboratory (due to the opening of the laboratory door), the experimental data is adopted after 20 minutes of stable data.
As shown in Figure 28, the boundary condition of HVAC inlet 1 decreased from 294.06 K to 293.45 K (0.61 K); the boundary condition of HVAC inlet 2 dropped from 293.45 K to 293.35 K (0.1 K). The activation of the HVAC system reduced the average laboratory air temperature from 295.74 K to 295.58 K (0.16 K), and the temperature reduction by the HVAC offsets the heat generated by the operator.

![Diagram](image)

Figure 27: (a) Temperature and (b) airflow speed for case 3; and (c) temperature and (d) airflow speed for case 4.
Figure 28: HVAC system boundary conditions (temperature) change of (a) inlet 1 and (b) inlet 2.

With the comparison of Figure 28 (a) and Figure 28(b), this experiment reveals that the temperature reduction of inlet 2 is not as gradual as inlet 1, and the inlet 2 temperature decrease is small. The first reason could be that, as shown in Figure 29, inlet 1 and inlet 2 share a common channel. From the temperature sensors and airflow meter data, the temperature of inlet 2 is slightly higher, and the airflow velocity is slightly lower compared to inlet 1. This is because inlet 1 is closer to the “connection point”, and more air supply is delivered to inlet 1 (airflow pressure). The second reason could be that the measured temperature variation of inlet 2 is already less than the resolution range (± 0.3 K [105]) of the sensor and is not accurate relative to inlet 1.

The increase in temperature close to the CNC machine (Figure 27 [a] point D and Figure 27 [c] point D) indicates that the heat generated by the CNC machine affects the surrounding air temperature. Since point D is located to avoid the direct influence of the HVAC system (obscured by the CNC machine), it is less affected by the HVAC system and more accurately reflects the temperature around the CNC machine as shown in Figure 30.

Since the temperature decreases by 0.17 K, the width (0.84 m) and height (0.97 m) of the materials will change differently due to the variations in the coefficients of thermal expansion (see
Table 12 below). Inward shrinkage of the shape of the CNC machine due to the decrease in
temperature.

Table 12: Scenario 2 CNC machine deformation under different materials.

<table>
<thead>
<tr>
<th></th>
<th>Aluminum</th>
<th>Iron</th>
<th>Steel (hardened)</th>
<th>Stainless Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Width deformation (mm)</td>
<td>-0.00328</td>
<td>-0.00157</td>
<td>-0.00177</td>
<td>-0.00149</td>
</tr>
<tr>
<td>Height deformation (mm)</td>
<td>-0.00359</td>
<td>-0.00182</td>
<td>-0.00204</td>
<td>-0.00171</td>
</tr>
</tbody>
</table>

Figure 29: Airflow and the relative location of inlet 1 and inlet 2.

By comparing case 2 and case 4 vertically, different seasons and different HVAC system set temperatures affected the internal temperature of the S3 laboratory and the airflow temperature in the gap inlet under the same thermal generation conditions. Due to the seasonal changes, the temperature of the gap inlet increased by 1.84 K from case 2 (295.24 K) to case 3 (297.08 K) as shown in Table 5. The increase in the set temperature of the HVAC system resulted in four
validation temperature points in case 4 (Figure 27 [c]) higher than the four validation points in case 2 (Figure 20 [c]), respectively.

Figure 30: Point D obscured by the CNC machine.

4.2.2 Airflow Specification

Figure 25 (b) and Figure 25 (d) show the air velocity variation in the side view of the 2D plane. Point A displays the air velocity at 2.74 m from the door ($X = 2.74$ m); point B demonstrates the air velocity above the CNC machine; C and D indicate the air velocity near inlet 1; Point E illustrates the air velocity near the north wall (influenced by the air entering the laboratory from the gap inlet); and G and F show the air velocity at inlet 2. Points A to F are located in the laboratory as shown in appendix E.

The case 3 airflow velocity variation is from 0 m/s to 4.24 m/s with an error range of 0 m/s to 0.25 m/s. The case 4 airflow speed varies from 0 m/s to 4.3 m/s with an error range from 0.02 m/s to 0.35 m/s as shown in Table 13.
Table 13: Airflow speed simulation data, validation data, and error for case 3 and case 4.

<table>
<thead>
<tr>
<th></th>
<th>Simulation Data (m/s)</th>
<th>Validation Data (m/s)</th>
<th>Error (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Case 3</td>
<td>Case 4</td>
<td>Case 3</td>
</tr>
<tr>
<td>A</td>
<td>2.80</td>
<td>2.82</td>
<td>2.8</td>
</tr>
<tr>
<td>B</td>
<td>0.20</td>
<td>0.20</td>
<td>0</td>
</tr>
<tr>
<td>C</td>
<td>1.03</td>
<td>1.03</td>
<td>1</td>
</tr>
<tr>
<td>D</td>
<td>0.65</td>
<td>0.65</td>
<td>0.9</td>
</tr>
<tr>
<td>E</td>
<td>0.88</td>
<td>0.95</td>
<td>0.7</td>
</tr>
<tr>
<td>F</td>
<td>0.66</td>
<td>0.66</td>
<td>0.8</td>
</tr>
<tr>
<td>G</td>
<td>0.73</td>
<td>0.73</td>
<td>0.7</td>
</tr>
</tbody>
</table>

The comparison between case 3 and case 4 indicates that the entry of the operator and the turning on of the CNC machine do not affect the airflow velocity range, as shown in Figure 31. After comparing with the previous scenario, the indoor airflow distribution did not change significantly with the change of seasons. However, the indoor air temperature changed over time.

At point A (Figure 25 [b] and Figure 25 [d]), where there is no object obstruction for the gap inlet airflow, the simulated data of this experiment exhibits good consistent performance. In the places where the influence is greater near the wall, such as point D, it demonstrates a relatively large error (0.25 m/s for case 3 and 0.35 m/s for case 4). However, point D is not the point of concern in this experiment. The air velocity plane is highly consistent with the temperature plane, demonstrating an accurate result.
Figure 31: (a) Airflow streamline in volume rendering for case 3 and (b) airflow streamline in volume rendering for case 4.

4.3 Scenario 3: Case 5 and Case 6 CFD Results Comparison and Validation

Scenario 3 shows the variation in the temperature and airflow data throughout the experiment in a comparison of case 5 and case 6 from May 10, 2021 (temperature of S3 laboratory
The research direction of this experiment is to simulate the temperature and airflow variation in a factory environment. Since it is a factory environment, the entry and exit of operators will lead to the opening of laboratory doors. This experiment is conducted to investigate the effect of the operator on the internal environment of the laboratory with the door open. The original "gap inlet" has become the "door inlet".

4.3.1 Temperature Specification

With the door open, this experiment verifies the accuracy of the simulation using data from four detection points. As in the previous two sets of experiments, the data were averaged using on half hour of temperature data at each point. The locations of the four points are shown in appendix F.

Due to the door opening, the air from outside the S3 laboratory enters the laboratory at a lower speed (0.3 m/s). This air tended to be warmer than the laboratory air. As shown in Figure 32 (a), the heat for case 5 comes from the CNC machine, the hot air outside the door and the electric lamps. Case 6 adds the heat production of the operator compared to case 5 as shown in Figure 33. The whole door acted as an air inlet (door inlet), affecting the temperature distribution in the area close to the door. The air entering the laboratory from the door inlet created a triangle in the temperature distribution near the door as shown in Figure 32 (b). The triangular temperature distribution was formed because the cold air from inlet 2 influenced the upper part of the triangle. In contrast to scenario 1 and scenario 2, the door inlet in scenario 3 did not reach the north wall in a straight line, but formed a curved line as shown in Figure 32 (c). The larger air inlet region leads to higher temperatures near the doors than the previous two scenarios.

As Figure 34 (a) and Figure 34 (c) shows, the temperature range of case 5 is 294.4K to 300K and the temperature range of case 6 is 294.6K to 300.4K. The 2D plane view (Figure 35 [a]}
and Figure 35 [b]) of the CNC machine derives that the temperature distribution in the vicinity of the CNC machine changes when the operator enters the laboratory.

Figure 32: (a) Temperature contour, (b) temperature volume rendering, and (c) bottom view of volume rendering in case 5.
Figure 33: Temperature contour in case 6.

The percentage error for the four points are shown in Table 14. The average error of case 5 is 0.079%, while the average error of case 6 is 0.094%. The error in Case 6 is larger than the error in case 5 regardless of temperature range and temperature percentage error. This situation indicates that in complex cases, it increases the error of the simulation.

Table 14: Temperature percentage error of case 5 and case 6.

<table>
<thead>
<tr>
<th>Simulation data(K)</th>
<th>Validation data(K)</th>
<th>Percentage error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 5</td>
<td>Case 6</td>
<td>Case 5</td>
</tr>
<tr>
<td>A</td>
<td>295.89</td>
<td>296.61</td>
</tr>
<tr>
<td>B</td>
<td>296.48</td>
<td>296.74</td>
</tr>
<tr>
<td>C</td>
<td>296.20</td>
<td>296.61</td>
</tr>
<tr>
<td>D</td>
<td>296.47</td>
<td>296.88</td>
</tr>
</tbody>
</table>

When the operator enters the room, the heat generated by the operator raises the temperature near the CNC machine by 0.5 K (Figure 36 [a] point C vs Figure 36 [c] point C). In the simulation data, it is concluded that the temperature increased by 0.41 K, which shows the accuracy of the simulation.
Figure 34: (a) Temperature (294.4k to 300.0k) and (b) airflow speed (0 m/s to 4.468 m/s for case 5; (c) temperature (294.6k to 300.4 k) and (d) airflow speed (0m/s to 4.312m/s) for case 6.

Figure 35: (a) 2D vertical view of contour near CNC machine for case 5; (b) 2D vertical view of contour near CNC machine for case 6.

Since the temperature increased by 0.41 K, the width (0.84 m) and height (0.97 m) of the materials will change (at different rates) due to their respective coefficients of thermal expansion. The different variations are shown in Table 15 below. The change in temperature causes the outward expansion of the frame of the CNC machine.
Table 15: Scenario 3 CNC machine deformation for different materials.

<table>
<thead>
<tr>
<th></th>
<th>Aluminum</th>
<th>Iron</th>
<th>Steel (hardened)</th>
<th>Stainless Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Width deformation</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(mm)</td>
<td>0.00792</td>
<td>0.00379</td>
<td>0.004271</td>
<td>0.00358</td>
</tr>
<tr>
<td><strong>Height deformation</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(mm)</td>
<td>0.00914</td>
<td>0.00437</td>
<td>0.00492</td>
<td>0.00413</td>
</tr>
</tbody>
</table>

Figure 36: (a) Temperature and (b) airflow speed for case 5; and (c) temperature and (d) airflow speed for case 6.
Due to the movement of operators, or ventilation requirements, many manufacturing industries work in both air-conditioned and naturally ventilated environments (scenario 3). Comparing scenario 1 in the door closing condition and scenario 3 in the door opening condition, the temperature differs significantly (3.81 °C = 3.81 K). Such a temperature variation can cause more significant errors in the working facility (Table 16).

Table 16: CNC machine deformation under different materials between opening and closing doors.

<table>
<thead>
<tr>
<th>Width deformation (mm)</th>
<th>Aluminum</th>
<th>Iron</th>
<th>Steel (hardened)</th>
<th>Stainless Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Width deformation (mm)</td>
<td>0.0736</td>
<td>0.0352</td>
<td>0.0397</td>
<td>0.0333</td>
</tr>
<tr>
<td>Height deformation (mm)</td>
<td>0.0850</td>
<td>0.0407</td>
<td>0.0458</td>
<td>0.0384</td>
</tr>
</tbody>
</table>

As shown in Table 16, a temperature variation of 3.81°C can cause structural (frame) changes to the CNC machine due to the coefficient of thermal expansion. Since this experiment investigates the result of changes to the frame from temperature variations around the CNC machine, this experiment does not address the changes inside the CNC machine (e.g., workpiece and cutting tool thermal expansion) due to thermal expansion. Take CNC machine drilling as an example. As shown in Figure 37 (a) and (b) the change in height of the CNC machine frame can affect the Z axis of the internal table of the CNC machine, which will eventually affect the accuracy of CNC machine. For example, due to the temperature change, the increase in the height of the CNC machine can lead to an increase in the distance between the cutting tool and the workpiece material. Adding more considerations like the accuracy of the CNC machine (0.01 mm [106]) could increase the margin of error (± 0.01 mm). By considering both the change in height and the machine’s accuracy, the typical tolerance of CNC machined parts can be reached (± 0.127 mm
and result in product scrap. Additionally, variation in width can lead to changes in the machine's X axis and ultimately to changes in the drill bit's position. Therefore, high precision equipment like CNC machines is required to accurately monitor the variation in the surrounding environment and make timely changes to improve precision.

Figure 37: (a) Height and width of the CNC machine and (b) internal view of the CNC machine in coordinate system.
4.3.2 Airflow Specification

Due to the opening of the laboratory door, airflow entry from outside the laboratory is slowed down, as shown in Figure 38. As a result, the airflow entering through the door does not reach the north wall as it could in the previous two comparison experiments. This experiment investigates the airflow speed and airflow trajectory in the vicinity of the CNC machine. However, in this scenario, due to the accuracy limitation of the experimental equipment, the measured data for multiple points are zero (A, B and E) and have no argumentative value (as show in Figure 38, it can be seen that a large area of the experiment is in a pink zone with low airflow velocity). The airflow part is not discussed in this experiment. For future research, the comparative results of airflow at various points can be continued by improving the resolution of the airflow meter.

Figure 38: (a) Airflow streamline in volume rendering for case 5 and (b) airflow streamline in volume rendering for case 6.

4.4 Limitations and Discussion

The current CFD simulation requires much time for calculation. In the author's computer (MSI PC with Intel(R) Core (TM) i7-10750H CPU @ 2.60GHz 2.59 GHz and 16.0 GB RAM), for example, the time required to simulate the laboratory situation once is about three hours, which means that the temperature and airflow in the laboratory three hours ago can be known after three
hours of simulation. For future research, the computer's computing power be improved to get real-time variations in the laboratory within a reasonable timeframe.

When the computing capacity is powerful enough, the manufacturing industry can establish a digital twin and can get real-time data remotely to detect factory conditions. While more metadata is populated (with different kinds of sensors), manufacturing can create a more sophisticated virtual cloud system that can accurately detect the physical object of the manufacturing factory and simulate the various processes that occur inside the manufacturing industry (such as changes in temperature and humidity during the production of a product, which may cause the product to be ruined). With the digital twin cloud system in operation, the manufacturing industry could investigate performance issues (such as testing the actual effective utilization of the HVAC system) and transmitting data to the manufacturing cloud system to coordinate the management of the factory’s performance (improving the efficiency of the air conditioning system or even predicting the life cycle of the air conditioning system).

Since the methodology of this experiment is based on a theoretical analysis, the pros and cons are analyzed using theoretical knowledge. Nevertheless, no other methods were attempted that could also theoretically implement the simulation. The surface temperature for this experiment was measured by a thermal imager and calculated using equations. In the future, when more thermal generators are considered, the objects in the laboratory can be used as a heat source with constant energy output instead of the measured values, which can make the experimental acquisition process less difficult and less time intensive.

The unstructured mesh method was applied in the meshing step due to the non-linear variation of temperature, air velocity (various types of shapes or functions), and the complex geometric appearance (including more complex potential heat-producing objects in future
research. In proximity to the object, the experiment could use an inflation layer to create a prism mesh because the mesh near the object inflates and grows normal to the object's surface. In this way, research can create a rectangular mesh and capture deep gradients of temperature and airflow variables normal to the surface.
Chapter 5: Conclusion and Future Research

This experiment revealed that using CFD technology provided accurate results for modeling the air temperature and airflow in the machining laboratory. The research found that the CFD technique has apparent advantages for indoor environment simulation, with an average error of 0.10% (minimum percentage error 0.079 % and maximum percentage error 0.15 %) for temperature and an error range of 0 m/s - 0.65 m/s for airflow velocity. At the same time, it can accurately reflect the impact of operator entry and exit on temperature changes near CNC machines, which verifies the correctness of the theoretical approach and analytical model. Even when HVAC system varies with season and set temperature, these data demonstrate the accuracy of this experiment in simulating temperature and airflow in different seasons and indicate that the ANSYS Fluent model is correctly chosen for this experiment. During temperature variations, the width (maximum change value: 0.0736 mm) and height (maximum change value 0.0850 mm) of the CNC machine frame are expected to change due to thermal expansion, which in turn can affect the accuracy of the CNC machine. This research also shows the importance of accurate monitoring of the manufacturing environment. CFD simulations can detect slight changes that the thermostat for HVAC systems cannot readily detect. If these temperature changes can be transmitted to the air conditioning system, the laboratory temperature can be adjusted in time and reduce the error caused by the temperature. At the same time, airflow is essential for thermal comfort. CFD simulation data variation is utilized to provide reference data to the HVAC system. This provides a solution to improve the internal environment of the laboratory and enhance thermal comfort.
Based on the accuracy of the simulation data, this experiment can be reasonably extended to the manufacturing industry. For example, in semiconductor manufacturing, research has found a relationship between the change in temperature in the working area and the type, form, size, quantity and location of contaminated particles [108]. Furthermore, these contaminant particles gathered on the EUV pellicle will eventually impact the lifetime of the EUV. The EUV system needs stringent environmental monitoring to produce chip fabrication that is below 7nm. CFD simulation technique can provide a more accurate method of detecting environmental changes to benefit the manufacturing industry.

In all, accurate simulations of the ambient conditions can provide the basis for more precise manufacturing and additional data to investigate thermal comfort inside the manufacturing factory. Future research is recommended to include the variation of airflow vectors in each direction (X, Y and Z axis), the addition of air humidity conditions, and the effect on the results after the addition of hazardous gas conditions. CFD technology can also be used to improve HVAC system design like change the location and direction of HVAC system inlets to provide better airflow to manufacturing factories. When CFD technology combines with digital twin technology, more accurate monitoring and predictions can be made.
References


[91] “Module 3: Global Mesh Controls Introduction to ANSYS Meshing.”


[105] “Amazon.Com: Govee Bluetooth Hygrometer Thermometer, Indoor Temp Humidity Sensor with Notification Alert & 2s Refresh Speed, 2 Years Data Storage Export, Suitable for Room Greenhouse Wine Cellar Basement : Patio, Lawn & Garden” [Online]. Available: https://www.amazon.com/dp/B087313N8F/ref=sspa_dk_detail_3?psc=1&pd_rd_i=B087313N8F&pd_rd_w=yrvoX&pf_rd_p=887084a2-5c34-4113-a4f8-b7947847c308&pd_rd_wg=lbZxi&pf_rd_r=Y5G32CWTRCKKS4S8R8Y9&pd_rd_r=2ab1371-0974-4694-be6f-18e6aa1155ad&spLa=ZW5jcnlwdGVkUXVhBGlmaWVyPUEzRjlCWVdQVkkNFTFFQlVuY3J5cHRlZElkPUEwNzYzOTIzMiBQVEpLNVpGTjNXTSZlbmNyeXB0ZWRBZElkPUEwMjkyMTAxM041VlhHTk3MEY0UyZ3aWRnZXROYW1lNXNwX2RldGFpbCZhY3Rpb249Y2xpY2tSZWRpcmVjdCZkb05vdExvZ0NsaWNRydWU=.

[106] Panel, O., “Emergency Stop Spindle and Feed Override 11 Pot Tool Changer Access Door Coolant Tank Base Stand.”


## Appendix A: Element Quality Table

Table A1: Element quality table

<table>
<thead>
<tr>
<th>Object Name</th>
<th>Geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>State</td>
<td>Fully Defined</td>
</tr>
<tr>
<td>Definition</td>
<td></td>
</tr>
<tr>
<td>Source</td>
<td>C:\Users\yang7\Desktop\Thesis paper\5 13 closed door</td>
</tr>
<tr>
<td></td>
<td>second collect data\closed door ansys\closed</td>
</tr>
<tr>
<td></td>
<td>door_files\dp0\FFF\DM\FFF.agdb</td>
</tr>
<tr>
<td>Type</td>
<td>DesignModeler</td>
</tr>
<tr>
<td>Length Unit</td>
<td>Meters</td>
</tr>
<tr>
<td>Bounding Box</td>
<td></td>
</tr>
<tr>
<td>Length X</td>
<td>7.5438 m</td>
</tr>
<tr>
<td>Length Y</td>
<td>3.302 m</td>
</tr>
<tr>
<td>Length Z</td>
<td>4.2672 m</td>
</tr>
<tr>
<td>Properties</td>
<td></td>
</tr>
<tr>
<td>Volume</td>
<td>100.51 m³</td>
</tr>
<tr>
<td>Scale Factor Value</td>
<td>1.</td>
</tr>
<tr>
<td>Statistics</td>
<td></td>
</tr>
<tr>
<td>Bodies</td>
<td>1</td>
</tr>
<tr>
<td>Active Bodies</td>
<td>1</td>
</tr>
<tr>
<td>Nodes</td>
<td>425483</td>
</tr>
<tr>
<td>Elements</td>
<td>2266996</td>
</tr>
<tr>
<td>Mesh Metric</td>
<td>Element Quality</td>
</tr>
<tr>
<td>Min</td>
<td>0.209139434952001</td>
</tr>
<tr>
<td>Max</td>
<td>0.999941298123163</td>
</tr>
<tr>
<td>Average</td>
<td>0.836805736425161</td>
</tr>
<tr>
<td>Standard Deviation</td>
<td>9.51574443608371E-02</td>
</tr>
<tr>
<td>Update Options</td>
<td></td>
</tr>
<tr>
<td>Assign Default Material</td>
<td>No</td>
</tr>
<tr>
<td>Basic Geometry Options</td>
<td></td>
</tr>
<tr>
<td>Parameters</td>
<td>Independent</td>
</tr>
<tr>
<td>Parameter Key</td>
<td>Yes</td>
</tr>
<tr>
<td>Attributes</td>
<td>Yes</td>
</tr>
<tr>
<td>Attribute Key</td>
<td>Yes</td>
</tr>
<tr>
<td>Named Selections</td>
<td>Yes</td>
</tr>
<tr>
<td>Named Selection Key</td>
<td>Yes</td>
</tr>
<tr>
<td>Material Properties</td>
<td>Yes</td>
</tr>
<tr>
<td>Advanced Geometry Options</td>
<td></td>
</tr>
<tr>
<td>Use Associativity</td>
<td>Yes</td>
</tr>
<tr>
<td>Coordinate Systems</td>
<td>Yes</td>
</tr>
<tr>
<td>Coordinate System Key</td>
<td></td>
</tr>
<tr>
<td>-------------------------------------------</td>
<td>-------</td>
</tr>
<tr>
<td>Reader Mode Saves Updated File</td>
<td>No</td>
</tr>
<tr>
<td>Use Instances</td>
<td>Yes</td>
</tr>
<tr>
<td>Smart CAD Update</td>
<td>Yes</td>
</tr>
<tr>
<td>Compare Parts On Update</td>
<td>No</td>
</tr>
<tr>
<td>Analysis Type</td>
<td>3-D</td>
</tr>
<tr>
<td>Import Facet Quality</td>
<td>Source</td>
</tr>
<tr>
<td>Clean Bodies On Import</td>
<td>No</td>
</tr>
<tr>
<td>Stitch Surfaces On Import</td>
<td>None</td>
</tr>
<tr>
<td>Decompose Disjoint Geometry</td>
<td>Yes</td>
</tr>
<tr>
<td>Enclosure and Symmetry Processing</td>
<td>No</td>
</tr>
</tbody>
</table>
## Appendix B: Aspect Ratio Table

Table B1: Aspect ratio table

<table>
<thead>
<tr>
<th>Object Name</th>
<th>Geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>State</td>
<td>Fully Defined</td>
</tr>
</tbody>
</table>

### Definition

<table>
<thead>
<tr>
<th>Source</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>C:\Users\yang7\Desktop\Thesis paper\5 13 closed door second collect data\closed door ansys\closed door_files\dp0\FFF\DM\FFF.agdb</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Type</th>
<th>DesignModeler</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Length Unit</th>
<th>Meters</th>
</tr>
</thead>
</table>

### Bounding Box

<table>
<thead>
<tr>
<th>Length X</th>
<th>7.5438 m</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length Y</td>
<td>3.302 m</td>
</tr>
<tr>
<td>Length Z</td>
<td>4.2672 m</td>
</tr>
</tbody>
</table>

### Properties

<table>
<thead>
<tr>
<th>Volume</th>
<th>100.51 m³</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Scale Factor Value</th>
<th>1.0</th>
</tr>
</thead>
</table>

### Statistics

<table>
<thead>
<tr>
<th>Bodies</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Active Bodies</td>
<td>1</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Nodes</th>
<th>425483</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elements</td>
<td>2266996</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Mesh Metric</th>
<th>Aspect Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Min</td>
<td>1.1617</td>
</tr>
<tr>
<td>Max</td>
<td>10.927</td>
</tr>
<tr>
<td>Average</td>
<td>1.84707853794184</td>
</tr>
</tbody>
</table>

| Standard Deviation | 0.459852755299415 |

### Update Options

| Assign Default Material | No |

### Basic Geometry Options

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Independent</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameter Key</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Attributes</th>
<th>Yes</th>
</tr>
</thead>
</table>

| Attribute Key | |

<table>
<thead>
<tr>
<th>Named Selections</th>
<th>Yes</th>
</tr>
</thead>
</table>

| Named Selection Key | |

| Material Properties | Yes |

### Advanced Geometry Options

<table>
<thead>
<tr>
<th>Use Associativity</th>
<th>Yes</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Coordinate Systems</th>
<th>Yes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coordinate System Key</td>
<td></td>
</tr>
<tr>
<td>-----------------------</td>
<td>--</td>
</tr>
<tr>
<td>Reader Mode Saves Updated File</td>
<td>No</td>
</tr>
<tr>
<td>Use Instances</td>
<td>Yes</td>
</tr>
<tr>
<td>Smart CAD Update</td>
<td>Yes</td>
</tr>
<tr>
<td>Compare Parts On Update</td>
<td>No</td>
</tr>
<tr>
<td>Analysis Type</td>
<td>3-D</td>
</tr>
<tr>
<td>Import Facet Quality</td>
<td>Source</td>
</tr>
<tr>
<td>Clean Bodies On Import</td>
<td>No</td>
</tr>
<tr>
<td>Stitch Surfaces On Import</td>
<td>None</td>
</tr>
<tr>
<td>Decompose Disjoint Geometry</td>
<td>Yes</td>
</tr>
<tr>
<td>Enclosure and Symmetry Processing</td>
<td>No</td>
</tr>
</tbody>
</table>
Appendix C: Location of Temperature Data Collection Point A to D in Machining Laboratory for Scenario 1

Table C1: Location of temperature data collection point A to D in machining laboratory for scenario 1

<table>
<thead>
<tr>
<th></th>
<th>X (m)</th>
<th>Y (m)</th>
<th>Z (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Case 1</td>
<td>Case 2</td>
<td>Case 1</td>
</tr>
<tr>
<td>A</td>
<td>6.2</td>
<td>6.2</td>
<td>1.57</td>
</tr>
<tr>
<td>B</td>
<td>2.65</td>
<td>2.65</td>
<td>1.57</td>
</tr>
<tr>
<td>C</td>
<td>4.24</td>
<td>4.24</td>
<td>1.95</td>
</tr>
<tr>
<td>D</td>
<td>0.25</td>
<td>3.8</td>
<td>0</td>
</tr>
</tbody>
</table>

Note: X represents the length; Y represents the height; Z represents the width.

Table C.1 Temperature data collection point A to D for scenario 1.
Appendix D: Location of Temperature Data Collection Point A to D in Machining Laboratory for Scenario 2

Table D1: Location of temperature data collection point A to D in machining laboratory for scenario 2

<table>
<thead>
<tr>
<th></th>
<th>X (m)</th>
<th>Y (m)</th>
<th>Z (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>6.22</td>
<td>1.57</td>
<td>1.47</td>
</tr>
<tr>
<td>B</td>
<td>2.72</td>
<td>1.57</td>
<td>1.48</td>
</tr>
<tr>
<td>C</td>
<td>4.24</td>
<td>1.95</td>
<td>2.27</td>
</tr>
<tr>
<td>D</td>
<td>3.8</td>
<td>1.22</td>
<td>3.47</td>
</tr>
</tbody>
</table>

Table D.1 Temperature data collection point A to D for scenario 2.
Appendix E: Location of Airflow Data Collection Point A to G in Machining Laboratory for Scenario 2

Table E1: Location of airflow data collection point A to G in machining laboratory for scenario 2

<table>
<thead>
<tr>
<th></th>
<th>X</th>
<th></th>
<th>Y</th>
<th></th>
<th>Z</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Case 3</td>
<td>Case 4</td>
<td>Case 3</td>
<td>Case 4</td>
<td>Case 3</td>
<td>Case 4</td>
</tr>
<tr>
<td>A</td>
<td>2.74</td>
<td>2.74</td>
<td>0</td>
<td>0</td>
<td>2.04</td>
<td>2.04</td>
</tr>
<tr>
<td>B</td>
<td>4.23</td>
<td>4.23</td>
<td>1.86</td>
<td>1.86</td>
<td>3.41</td>
<td>3.41</td>
</tr>
<tr>
<td>C</td>
<td>6.20</td>
<td>6.20</td>
<td>2.39</td>
<td>2.39</td>
<td>0.7</td>
<td>0.7</td>
</tr>
<tr>
<td>D</td>
<td>7.22</td>
<td>7.22</td>
<td>2.39</td>
<td>2.39</td>
<td>1.48</td>
<td>1.48</td>
</tr>
<tr>
<td>E</td>
<td>7.42</td>
<td>7.42</td>
<td>0</td>
<td>0</td>
<td>2.04</td>
<td>2.04</td>
</tr>
<tr>
<td>F</td>
<td>1.14</td>
<td>1.14</td>
<td>2.39</td>
<td>2.39</td>
<td>1.48</td>
<td>1.48</td>
</tr>
<tr>
<td>G</td>
<td>2.74</td>
<td>2.74</td>
<td>2.39</td>
<td>2.39</td>
<td>0.43</td>
<td>0.43</td>
</tr>
</tbody>
</table>

Table E.1 Airflow data collection point A to G for scenario 2.
## Appendix F: Location of Temperature Data Collection Point A to D in Machining Laboratory for Scenario 3

Table F1: Location of temperature data collection point A to D in machining laboratory for scenario 3

<table>
<thead>
<tr>
<th></th>
<th>Case 5</th>
<th>Case 6</th>
<th>Case 5</th>
<th>Case 6</th>
<th>Case 5</th>
<th>Case 6</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>6.22</td>
<td>6.22</td>
<td>1.57</td>
<td>1.57</td>
<td>1.47</td>
<td>1.48</td>
</tr>
<tr>
<td>B</td>
<td>2.72</td>
<td>2.72</td>
<td>1.57</td>
<td>1.57</td>
<td>1.48</td>
<td>1.48</td>
</tr>
<tr>
<td>C</td>
<td>4.24</td>
<td>4.24</td>
<td>1.95</td>
<td>1.95</td>
<td>2.27</td>
<td>2.27</td>
</tr>
<tr>
<td>D</td>
<td>3.8</td>
<td>3.8</td>
<td>1.22</td>
<td>1.22</td>
<td>3.47</td>
<td>3.47</td>
</tr>
</tbody>
</table>

Note: X represents the length; Y represents the height; Z represents the width.