The Numerical Investigation of the Effects of Sand Ingestion on Compressor Blade Erosion

Taha İrfan Çağdaş

Thesis submitted to the faculty of the Virginia Polytechnic Institute and State University in partial fulfillment of the requirements for the degree of Master of Science

In
Aerospace Engineering

Luca Massa, Chair
K.Todd Lowe
Kevin G.Wang

December 13, 2023
Blacksburg, Virginia

Keywords: multiphase flow, compressor blade erosion, sand ingestion, numerical methods
The Numerical Investigation of the Effects of Sand Ingestion on Compressor Blade Erosion

Taha İrfan Çağdaş

ABSTRACT

The performance of aircraft engines can be significantly affected by the variety of foreign particles that are mixed into the air while operating under miscellaneous conditions. In particular, aircraft engines that operate in sandy or dusty conditions may fail within minutes of exposure to particle-laden flow due to foreign particle deposition on hot section components or erosion occurring on the compressor and turbine blades.

For these reasons, the effect of sand ingestion on erosion, which may occur in the turbine and compressor blades, was studied in this master’s thesis. In this master’s thesis, the effect of sand ingestion on erosion on the M250 turboshaft engine’s compressor blades will be investigated with the aid of numerical methods. In this study, we used the OpenFOAM software to solve the multiphase flow problem from the standpoint of finite control methods and the Eulerian-Lagrangian framework. The initial sand distribution conditions were taken from the Ph.D. thesis written by Olshefski, K. T. (2023) [1]. The compressor blade was modeled as 2D, which has a NACA 6510 profile shape, with a chord length of 63 mm.

The results show that the leading edge and the suction side of the compressor, i.e. the upper half of the compressor, eroded more compared to the trailing edge, and the pressure side. Results also show that as the sand particle distribution becomes non-uniform the most eroded region shifts toward the trailing edge. In addition, for varying angles of attack, the region where the erosion occurs alters periodically. We observed that as the angle of attack increases, the eroded region shifts toward the trailing edge, but when the angle of attack is kept increasing the eroded region shifts back to the leading edge again.

In conclusion, the non-uniformity of sand particle loading has a strong effect on the determination of the eroded regions. Furthermore, the variation of the angle of attack has a huge role in both the determination of eroded regions and the amount of eroded material.
The Numerical Investigation of the Effects of Sand Ingestion on Compressor Blade Erosion

Taha İrfan Çağdaş

GENERAL AUDIENCE ABSTRACT

In this master’s thesis, the effect of sand ingestion on compressor blade erosion was investigated with the help of numerical methods. The compressor is one of the vital parts of air-breathing engines such as turboshaft, turbofan, turbojet, and turboprop engines. Therefore, the erosion on the compressor blades may cause pressure surges, which could cause severe problems in the operation of aircraft or airplanes operating under dusty conditions.

Historically, it is reported that a TransAmerican aircraft propelled by Alison T-56 engines lost two of its four engines after 3 to 4 minutes of exposure to volcanic ash while flying over Mt. St. Helens in 1980. Another example of the effects of sand ingestion is an MV-22 Osprey crash that happened during a training exercise in Hawaii, claiming the lives of two US Marines and injuring twenty other personnel in 2015. It was attributed that the cause of the fatal accident was the ingestion of dust that caused engine failure.

Therefore, our intention in studying this field is to have an understanding of the regions of compressor blades that are vulnerable to erosion.

In this master’s thesis, numerical methods based on the finite volume method were used to obtain numerical solutions to estimate erosion on the compressor blade by utilizing OpenFOAM. We would like to recommend a nice OpenFOAM tutorial for those who are interested in applying numerical methods using OpenFOAM, taught by Jozsef Nagy accessible on YouTube, https://www.youtube.com/@OpenFOAMJozsefNagy.

Also, for creating geometry and mesh generation of an airfoil for the use of OpenFOAM, we would like to recommend the tutorial presented by Ali Ikhsanul, accessible on YouTube via this link https://www.youtube.com/@aliikhsanul7982.

These tutorial videos could help those who are interested in Openfoam but do not have much experience with Openfoam.

The work in this master’s thesis indicates that the leading edge of the compressor blade is more prone to be eroded than the trailing edge. In addition, it is shown that the eroded region distribution is highly dependent on the angle of attack of sand particles.
Dedicated to my mother, my father, and my sister
Acknowledgment

First and foremost, I would like to thank the Ministry of National Education, Republic of Turkey officials, who awarded me the scholarship that allowed me to pursue my master’s degree in aerospace engineering at Virginia Polytechnic Institute and State University. Thanks to financial support from the Ministry of National Education, Republic of Turkey, I have been able to complete my master’s degree. I would also like to thank my family members for their unwavering support. I also would like to express my heartfelt appreciation and gratitude to my advisor, Dr. Luca Massa, for his support, his trust in me, and guidance in my master’s education journey. It was a blessing for me to conduct my master’s thesis study under the supervision of Dr. Massa. Dr. Massa significantly helped me to develop an interest in numerical methods and unlocked a wealth of knowledge related to multiphase flow physics that I had not been aware of before. I also would like to thank Dr. Kevin Wang and Dr. Todd Lowe for their support in my master’s thesis. I also would like to offer my genuine gratitude to Dr. Kristopher Olshefski for his generous help during my time at Virginia Tech.

December, 2023
Taha İrfan Çağdaş
# Table of Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>List of Figures</td>
<td>viii</td>
</tr>
<tr>
<td>List Of Tables</td>
<td>xi</td>
</tr>
<tr>
<td>1. Introduction</td>
<td>1</td>
</tr>
<tr>
<td>1.1 Contributions of the Present Research</td>
<td>3</td>
</tr>
<tr>
<td>2. A Brief Introduction to Multiphase Flows</td>
<td>4</td>
</tr>
<tr>
<td>2.1 Volumetric Fraction of Dispersed Phase</td>
<td>5</td>
</tr>
<tr>
<td>2.2 Stokes Number of the Particles</td>
<td>7</td>
</tr>
<tr>
<td>2.3 Multiphase Flow Coupling</td>
<td>8</td>
</tr>
<tr>
<td>3. Theory</td>
<td>14</td>
</tr>
<tr>
<td>3.1 Governing Equations of Single-phase Flow</td>
<td>14</td>
</tr>
<tr>
<td>3.2 Governing Equations of Multiphase Flow</td>
<td>18</td>
</tr>
<tr>
<td>3.2.1 Governing Equations for Eulerian-Eulerian Approach</td>
<td>19</td>
</tr>
<tr>
<td>3.2.2 Governing Equations for Eulerian-Lagrangian Approach</td>
<td>21</td>
</tr>
<tr>
<td>3.3 Forces Acting on a Single Particle</td>
<td>22</td>
</tr>
<tr>
<td>3.3.1 Drag Force</td>
<td>22</td>
</tr>
<tr>
<td>3.3.2 Body Forces</td>
<td>24</td>
</tr>
<tr>
<td>3.3.3 Other Forces</td>
<td>24</td>
</tr>
<tr>
<td>3.4 Erosion Models</td>
<td>26</td>
</tr>
<tr>
<td>3.4.1 Finnie Model</td>
<td>26</td>
</tr>
<tr>
<td>3.4.2 Neilson and Gilchrist Model</td>
<td>27</td>
</tr>
<tr>
<td>3.4.3 Zhang Model</td>
<td>28</td>
</tr>
<tr>
<td>4. Experimental Setup</td>
<td>29</td>
</tr>
<tr>
<td>4.1 Test Cell</td>
<td>29</td>
</tr>
<tr>
<td>4.2 Anisokinetic Probe</td>
<td>31</td>
</tr>
<tr>
<td>4.3 An Estimation for the Chord Length of the Compressor Blade</td>
<td>32</td>
</tr>
<tr>
<td>4.4 Particle Visualization by Illuminated Scattering</td>
<td>33</td>
</tr>
<tr>
<td>5. Numerical Solution to the Multiphase Flow</td>
<td>38</td>
</tr>
<tr>
<td>5.1 OpenFOAM Overview</td>
<td>38</td>
</tr>
<tr>
<td>5.2 The “simpleReactingParcelFoam” Solver</td>
<td>39</td>
</tr>
</tbody>
</table>
5.3 The “simpleReactingParcelFoam” Tutorial .......................................................... 42
5.4 Introducing “sand” as a Solid Species to OpenFOAM ....................................... 43
5.5 Geometry Creation and the Mesh Generation ...................................................... 46
5.6 The “0” Directory of the Tutorial Folder .......................................................... 50
5.7 The “constant” Directory of the Tutorial Folder ................................................. 52
5.8 The “system” Directory of the Tutorial Folder ................................................ 55
6. Results and Discussion 56
   6.1 Interpretation of Eroded Volume Results for Uniform Sand Distribution Case ...... 57
   6.2 Interpretation of Eroded Volume Results for Non-uniform Sand Distribution Case ..... 61
   6.3 The Comparison of Uniform and Non-uniform Sand Loading Case .................... 65
   6.4 The Evaluation of Eroded Volume for Different Angles of Attack In the Case of Stationary Compressor Blade .......................................................... 67
   6.5 Mesh Refinement Analysis ............................................................................. 71
7. Summary 72
8. Conclusion 75
References 77
List of Figures

1 Introduction 1

2. A Brief Introduction to Multiphase Flow 4

Figure 2.1: Particle-laden flow classification map. (Taken from Elghobashi S. (2006) [6], with author’s permission.) .................................................................10

3. Theory 14

4. Experimental Setup 28

Figure 4.1: The mass concentration date of sand particles both for high loading, is on the right, and low loading conditions, is on the left. Taken from Olshefski K.T. (2023) [1], with author’s permission. ..................................................34

Figure 4.2: Volumetric fraction distribution on the left, Number of particles per second on the right for 560 different inlet conditions. ........................................................................35

Figure 4.3: Volumetric fraction distribution on the left, Number of particles per second on the right for 35 different inlet conditions. .................................................................36

5. Numerical Solution to the Multiphase Flow 38

Figure 5.1: The part of the simpleReactingParcelFoam.C file where the equations are solved. ...40

Figure 5.2: The part of the createFields.H file that contains the information about variable addressing........................................................................................................41

Figure 5.3: The part of sand. H file that shows the entries that need to be changed. .................44

Figure 5.4: The part of sand. C file that shows the entries that need to be changed. .................45

Figure 5.5: The modification in the “files” file under the “Make” folder.................................46

Figure 5.6: The drawing of the compressor blade profile that has 4 different inlet conditions, each color represents different inlet conditions. .........................................................49

Figure 5.7: The “air” file located in “0” folder, entries from “BackFace” to “Inlet4” ...............51

Figure 5.8: The “reactingCloud1Properties” entries related to force and injection model..........53

Figure 5.9: Some functions that were used for obtaining some useful data. ..........................54

Figure 5.10: Some entries of “controlDict” file located in “system” directory. .........................55

6. Results and Discussion 56

Figure 6.1: Eroded volume of compressor blade suction side for the finest mesh level in uniform case. .........................................................................................................................58

Figure 6.2: Eroded volume of compressor blade pressure side for the finest mesh level in uniform case. .........................................................................................................................58
Figure 6.3: Eroded volume of compressor blade suction side for the medium mesh level in uniform case.

Figure 6.4: Eroded volume of compressor blade pressure side for the medium mesh level in uniform case.

Figure 6.5: Eroded volume of compressor blade suction side for the coarsest mesh level in uniform case.

Figure 6.6: Eroded volume of compressor blade pressure side for the coarsest mesh level in uniform case.

Figure 6.7: Eroded volume of compressor blade suction side for the finest mesh level in non-uniform case.

Figure 6.8: Eroded volume of compressor blade pressure side for the finest mesh level in non-uniform case.

Figure 6.9: Eroded volume of compressor blade suction side for the medium mesh level in non-uniform case.

Figure 6.10: Eroded volume of compressor blade pressure side for the medium mesh level in non-uniform case.

Figure 6.11: Eroded volume of compressor blade suction side for the coarsest mesh level in non-uniform case.

Figure 6.12: Eroded volume of compressor blade pressure side for the coarsest mesh level in non-uniform case.

Figure 6.13: Eroded volume of compressor blade suction side for the finest mesh level, Non-uniform sand mass loading case is on the left, Uniform sand loading case is on the right.

Figure 6.14: Number of the particle collisions on compressor blade suction side for the finest mesh level, Non-uniform sand mass loading case is on the left, Uniform sand loading case is on the right.

Figure 6.15: Eroded volume of compressor blade suction side for the coarsest mesh level in uniform case, 3° degrees of angle of attack is on the left, 6° degrees of angle of attack is on the right.

Figure 6.16: Eroded volume of compressor blade suction side for the coarsest mesh level in uniform case, 9° degrees of angle of attack is on the left, 12° degrees of angle of attack is on the right.

Figure 6.17: Eroded volume of compressor blade suction side for the coarsest mesh level in uniform case, 15° degrees of angle of attack is on the left, 18° degrees of angle of attack is on the right.

Figure 6.18: Eroded volume of compressor blade suction side for the coarsest mesh level in non-uniform case, 3° degrees of angle of attack is on the left, 6° degrees of angle of attack is on the right.
Figure 6.19: Eroded volume of compressor blade suction side for the coarsest mesh level in non-uniform case, 90 degrees of angle of attack is on the left, 120 degrees of angle of attack is on the right.

Figure 6.20: Eroded volume of compressor blade suction side for the coarsest mesh level in non-uniform case, 150 degrees of angle of attack is on the left, 180 degrees of angle of attack is on the right.

Figure 6.21: Total eroded volume per unit span for 3 different mesh levels.

7. Summary

Figure 7.1: Total eroded volume under different conditions.

Figure 7.2: Number of collisions per unit area for non-uniform sand loading case and the finest mesh, 0 degrees of angle of attack case is on the left, 9 degrees of angle of attack case is on the right.

Figure 7.3: Number of collisions per unit area for non-uniform sand loading case and the finest mesh, 0 degrees of angle of attack case is on the left, 9 degrees of angle of attack case is on the right.

8. Conclusion
List Of Tables

Table 2.1: Examples of single, multi-component, and multiphase flows ............................................ 4
1. Introduction

In this thesis, we will investigate the effects of sand particle-laden flow on the erosion of the compressor blade, with the help of numerical methods. For this investigation, we will utilize the OpenFOAM software, which is an open-source software written in the C++ environment. OpenFOAM software is able to solve the governing equations of the flow with the help of finite volume methods whose details can be elaborated on in Chapter 3.

In the scope of this master’s thesis, we will be interested in an isolated single compressor blade, by neglecting the influence of adjacent blades. We will also consider that the blade is stationary so that we would be able to neglect the variation of the angle of attack owing to the compressor blade rotation. Furthermore, we consider that the sand particle deposition on the compressor blade, and the phase changes would not occur in the scope of this work.

Our assumption that the sand particle deposition would not occur is based on the work done by Dunn M. G. (2012). According to Dunn M. G. (2012), particle deposition is not the essential damage mechanism if the turbine inlet temperature is below 1283 K [2]. In this thesis, we will investigate the erosion mechanism on the compressor blade of the Rolls-Royce Model 250-C20B turboshaft engine as a model. It is stated that for the Rolls-Royce Model 250-C20B turboshaft engine, the turbine inlet temperature does not exceed 1283 K [3]. Therefore, we can neglect the sand particle deposition on the compressor blade.

In Chapter 2, we will discuss the behavior of multiphase flow depending on the Stokes number and volumetric fraction rate. We will show that these 2 dimensionless numbers affect the turbulence formation, the forces acting on particles, and many other features of the flow.

In general, for 2 component gas-solid multiphase flow, the fluid is formed by the carrier medium and the dispersed phase. We will show that for our case, i.e. sand particle-laden airflow, sand particles can be considered by dispersed phase, and air can be considered as carrier medium. Therefore, throughout this thesis, we use the term “carrier medium” to refer to the airflow, and we will use the term “dispersed phase” to refer to sand particles. We will denote carrier medium, i.e. air, and dispersed phase, i.e. sand particles, with the subscripts “c” and “d” respectively. In general, we also denote the carrier medium velocity, i.e. airflow velocity with
“\( \mathbf{u} \)”, and we denote the dispersed particle velocity, i.e. sand particle velocity with “\( \mathbf{v} \). We use bold characters to denote vectors.

In Chapter 3, we will explain the theory and the governing equations of multiphase flow from both the consideration of the Eulerian-Eulerian and Eulerian-Lagrangian frameworks. We will explain some of the forces that act on the dispersed particles and will introduce some erosion models in the literature.

In Chapter 4, we will introduce an experimental setup which is located at the Advanced Power and Propulsion Laboratory (APPL) at Virginia Tech. We will focus on an experimental work conducted by Kristopher T. Olshefski under the supervision of Dr. Wing F. Ng and Dr. Todd K. Lowe. The reason for introducing this experimental work is the fact that we will utilize the results obtained from this experiment while determining the inlet sand particle distribution. This will help us identify the initial inlet conditions and the compressor blade geometry which provides us an important starting point for this study.

We will frequently refer to the work done by Olsheski K. T. (2023) [1], to explain our reasoning for inlet sand distribution or the choice of the chord length for the compressor blade. Therefore, in this thesis, we will frequently use the term “reference Ph.D. thesis” to give credit to Kristopher T. Olshefski’s Ph.D. dissertation.

In Chapter 5, we will briefly discuss how OpenFOAM can be utilized to solve the multiphase flow, how we can address the initial conditions, and how some minor adjustments can be made to modify the existing libraries of OpenFOAM to solve the flow problem that we are interested in.

In Chapter 6, we will present the results that we obtained from OpenFOAM and will discuss how we could interpret the results, and how the uniformity of sand flow and the variation of angle of attack affect the erosion.

In Chapter 7, we will summarize the results that are presented in Chapter 6 and will present a column graph that shows the relationship between eroded volume per unit span the uniformity of sand particles, and the angle of attack together.
In Chapter 8, we will conclude our work and we will point out that both uniformity and the angle of attack are substantial in terms of both determining the maximum eroded region of the blade and the total erosion.

1.1 Contributions of the Present Research

The effect of non-uniform particle distribution in the inflow on the erosion rates of compressor blades is not well understood. A new multicomponent flow model that allows to simulate sand particles clouds with non-uniform particle dispersion has been developed. The erosion rates on several three-dimensional airfoils have been analyzed using new distributions obtained at Virginia Tech. New insights on the correlation between particle distribution and the erosion rate maps are proposed. These have led to an improved understanding of the physics of erosion of compressor blades.
2. **A Brief Introduction to Multiphase Flows**

In this thesis, we are interested in sand particle-laden airflow which can be classified as a subcategory of what is known as multiphase multicomponent flow. Some examples of these kinds of flow can be given below,

<table>
<thead>
<tr>
<th></th>
<th>Single Phase</th>
<th>Multi-phase</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Single Component</strong></td>
<td>Water flow</td>
<td>The flow of water in the suction side of a pump in which water is under cavitation.</td>
</tr>
<tr>
<td><strong>Multi-Component</strong></td>
<td>Airflow</td>
<td>Sand particle-laden airflow</td>
</tr>
</tbody>
</table>

*Table 2.1: Examples of single, multi-component, and multiphase flows*

In practice, single-phase multi-component flows are treated as single-component flows with mean thermophysical properties. Such practices will be applicable if the molar weight of the components do not differ significantly, and the components of flow do not experience dissociation due to high temperature or condensation due to low temperature.

In the case of solid particle-contaminated flows, one could treat one of the phases as continuous and the other component as dispersed. To distinguish continuous and dispersed phases, we could say that one can pass from one point to another in the continuous phase while remaining in the same medium, while it is not possible to pass from one point to another without changing the medium within the dispersed phase.

Moreover, in a particle-laden airflow, it is evident that the airflow forms the continuous phase and solid particles form the dispersed phase. A test to determine if a phase is continuous or discrete can be achieved by considering the ratio of the distance between particles to the particles’ diameter.

In a continuous phase, one can determine the fluid properties at every point, and these properties should vary continuously. The density of a fluid at a local point can given as,

\[
\rho(x, y, z) = \lim_{\Delta V \to 0} \left( \frac{\Delta M}{\Delta V} \right)
\]

(2.1)
In Equation (2.1), $\rho$ denotes the density, $\Delta V$ denotes the volume, and $\Delta M$ denotes the mass. As the volume approaches zero, depending on where we are looking at the matter, we could either capture an atom or an absolute space between atoms. This results in surging values of density as the $\Delta V$ changes. However, when we increase $\Delta V$ volume adequately, the density of the matter enclosed within $\Delta V$ volume will converge the material density of the given matter. The $\Delta V$ volume that makes the density fluctuations of the matter insignificant, corresponds to the volume of $3.72 \times 10^{-21}$ m$^3$ for gaseous flows, which corresponds to the cube whose edge is around 150 nm [4]. In the numerical calculations of particle-laden flow, we will not divide the computational domain in a way that the edge of the domain would be less than 150 nm. Since we will not violate this condition, the flow of air can be safely regarded as a continuous flow.

### 2.1 Volumetric Fraction of Dispersed Phase

The volume fraction of the dispersed phase can be defined as the ratio of the volume of the particles to the mixture volume. Therefore, we can write,

$$\alpha_d = \frac{V_d}{V_m} \quad (2.2)$$

Similarly, we can write the volume fraction of the carrier phase as below,

$$\alpha_c = \frac{V_c}{V_m} \quad (2.3)$$

Since the volume of the mixture is formed by the volume of the carrier and the dispersed phase, we can write,

$$\alpha_c + \alpha_d = 1 \quad (2.4)$$

In which $\alpha$ denotes the volumetric fraction, $V$ denotes the volume, and subscripts c,d, and m denote carrier medium, dispersed phase, and mixture respectively.

We could also define densities related to both continuous and dispersed phases to ease the investigation of multiphase and multicomponent flow. We refer to the density of a matter that constitutes one of the components of the flow as material density. By denoting the material density of the components $\rho$ we can write the below expressions,

$$\rho_d = \frac{M_d}{V_d} \quad (2.5)$$
\[ \rho_c = \frac{M_c}{V_c} \] (2.6)

On the other hand, we could also define the bulk or apparent density of components such that bulk density represents the ratio of the mass of components to the total volume. Hence,

\[ \bar{\rho}_d = \frac{M_d}{V_m} \quad (2.7) \]

\[ \bar{\rho}_c = \frac{M_c}{V_m} \quad (2.8) \]

We can rewrite the bulk density expression in terms of volume fractions and the material density as below,

\[ \bar{\rho}_d = \alpha_d \rho_d \quad (2.9) \]

\[ \bar{\rho}_c = \alpha_c \rho_c \quad (2.10) \]

In the above equations, \( \rho \) denotes the material density and \( \bar{\rho} \) denotes the bulk density. Equation (2.9) and Equation (2.10) will be helpful for us to determine the volume fraction of the flow since we will use the bulk density data and the material densities of both air and sand particles from an experiment conducted at Virginia Tech as referred in [1].

In addition to the volume fraction of dispersed phase and carrier fluid, we could also look at the mass fraction of the components, which is a useful parameter in the consideration of possible chemical reactions between phases.

The mass ratio of the dispersed phase to the carrier fluid can be written as

\[ C = \frac{M_d}{M_c} = \frac{\bar{\rho}_d}{\bar{\rho}_c} = \frac{\alpha_d \rho_d}{\alpha_c \rho_c} \quad (2.11) \]

Likewise, for volume fraction, we could define mass fraction by proportioning the mass of the dispersed phase to the total mass multiphase flow. Hence, we can write,

\[ Y = \frac{M_d}{M_d + M_c} = \frac{\alpha_d \rho_d}{\alpha_d \rho_d + \alpha_c \rho_c} = \frac{C}{C+1} \quad (2.12) \]

In the above equations \( C \) denotes the ratio of mass of dispersed phase to mass of continuous phase and \( Y \) denotes the mass fraction i.e. the ratio of mass of the dispersed phase to total mass.
2.2 Stokes Number of the Particles

Stokes number is a dimensionless number that is defined as the ratio of particles’ response time to the characteristic time of the flow. Stokes number explains how rapidly particles start to move following the streamlines of the carrier flow. If the Stokes number is less than or equal to 1, the flow can be regarded as Stokesian flow, such that particles have enough time to mimic the fluctuation of the velocity of the carrier flow. In such conditions, particles move with carrier flow. As Stokes number increases, particles become more indifferent to the change of velocity fluctuations of carrier flow.

Stokes number can be obtained by considering the flow conditions of Stokesian flow. Stokesian flow is obtained when the relative Reynolds number of the carrier phase is too low. The relative Reynolds number of the continuous phase can be expressed as below,

\[ Re_r = \left( \frac{D_p |u - v| \rho_c}{\mu_c} \right) \]  \hspace{1cm} (2.13)

In which \( D_p \) is particle diameter, \( u \) is the carrier flow velocity, \( v \) is the dispersed phase velocity and \( \mu_c \) is carrier flow viscosity. If \( Re_r \ll 1 \), the flow becomes attached to the particle, and the flow around particles has fore and aft symmetry. In this case, it is possible to derive analytical solutions for the steady drag on particles. Stokes (1851) was the first to derive an expression for steady drag, and later, analytical expressions for drag forces were independently obtained by Hadamard J. S. (1911) and Rybczynski W. (1911) [5]. Hence, for the Stokesian flow, in which \( Re_r \ll 1 \) the force drag is given as below,

\[ F_D = 3\pi D(u - v)\mu_c \]  \hspace{1cm} (2.14)

By applying Newton’s second law we will write a relation between the acceleration of the particles and the drag force. Hence,

\[ m_d \frac{dv}{dt} = 3\pi D(u - v)\mu_c \]  \hspace{1cm} (2.15)

We can also write the mass of a single particle as below,

\[ m_d = \left( \frac{\pi D^3}{6} \right) \rho_d \]  \hspace{1cm} (2.16)

\[ \frac{D^2 \rho_d}{(18\mu_c)} \frac{dv}{dt} = (u - v) \]  \hspace{1cm} (2.17)
The coefficient of the acceleration which appears on the left-hand side of the Equation (2.17) is defined as momentum response time, or particle response time, and we will denote particle response time as $\tau_p$. Therefore, we can write,

$$\tau_p = \frac{D^2 \rho_d}{(18 \mu_c)}$$  \hspace{1cm} (2.18)

The solution of the differential equation (2.17) is given below,

$$v = u(1 - e^{-\frac{t}{\tau_p}})$$  \hspace{1cm} (2.19)

In which $v$ is the dispersed phase velocity vector, $u$ is the carrier medium velocity vector and $\tau_p$ is the particle response time. From Equation (2.19), we could say that the particle response time indicates the required time for a particle to reach 63% of the bulk flow velocity from at rest.

Stokes number is defined as the particle response time to the characteristic time of flow, which is denoted by $\tau_f$. We can write the characteristic time of flow as below,

$$\tau_f = \frac{L}{U}$$  \hspace{1cm} (2.20)

In which $L$ is the characteristic length of flow, which is, in our case, the length of the blade chord, which will be denoted by $c$ in the following equation. $U$ is the characteristic velocity of the flow, which can be chosen as the bulk velocity. Therefore, we can express the Stokes number as below,

$$St = \frac{D^2 \rho_d U}{18 \mu_c c \mu_c}$$  \hspace{1cm} (2.21)

### 2.3 Multiphase Flow Coupling

The term “Multiphase Flow Coupling” can be used for specifying if the dispersed particle flow affects the carrier medium flow, or if the particle-particle collision and interactions are important in the dispersed phase. Based on these considerations, one-way, two-way, and four-way coupled multiphase flows can be defined.

If the multiphase flow is considered a one-way coupled flow, this means that the carrier medium influences the dispersed phase flow, but the dispersed phase has no significant effect on
the properties of the carrier medium flow. In one-way coupled flows, the momentum and the energy are transferred from carrier flow to dispersed phase flow, so we could assume that the initial temperature of the carrier flow and the velocity of the carrier flow do not change due to the imbalance of carrier and dispersed flow temperatures or the velocities. In one-way coupled flows, temperature and the velocity of the dispersed phase asymptotically approach the temperature and the velocity of the carrier phase.

If the multiphase flow is considered a two-way coupled flow, dispersed phase particles also influence the properties of carrier medium flow properties as the carrier medium flow influences the dispersed phase particle. In these flows, the significant amount of heat transfer between the carrier and dispersed flow would change the temperature of the carrier flow thus, the velocity of the carrier flow is affected. These changes affect the way that the multiphase flow behaves.

If the multiphase flow is considered a four-way coupled flow, besides both carrier phase and the dispersed phase influence each other, the “four way-coupled flows” term also specifies that the particle-particle interactions are important. Based on the apparent density of particles, particle-particle interactions may be considered as either contact force-dominated or collision dominated.

As expected, one-way coupled flows can be comparatively easily handled compared to two-way or four-way coupled flows.

In the determination of the coupling of the multiphase flow, both the volumetric fraction of the dispersed phase and the Stokes number of particles are of paramount importance. Based on these parameters a flow classification chart was developed by Elghobashi S., (1994) and then updated by Elghobashi S., (2006) [6]. In Elghobashi S. (2006) work, the multiphase flow coupling was given in terms of volumetric fraction of the dispersed phase and the ratio of particle response time to Kolmogorov time scale. In another work in the literature which is done by Hoque M. M., et al. (2023), a classification map was given in terms of volumetric fraction of the dispersed phase and the Stokes number [7]. The flow classification map proposed by Elghobashi S. (2006) is given below,
Figure 2.1: Particle-laden flow classification map. (Taken from Elghobashi S. (2006) [6], with author’s permission.)

In the above chart, the -x axis shows the volume fraction of the dispersed phase, and the -y axis shows the ratio of particle response time to the Kolmogorov time scale. We showed that particle response time, $\tau_p$ is strongly related to the Stokes number, in Chapter (2.2).

Kolmogorov time scale, which is denoted by $\tau_K$ in Figure (2.1) related to the turbulent characteristics of the flow, which is given as,

$$\tau_K = \sqrt{\frac{\nu}{\epsilon}}$$  \hspace{1cm} (2.22)

In which $\nu$ denotes the carrier flow kinematic viscosity, $\epsilon$ denotes the dissipation rate of turbulent kinetic energy. For given conditions of characteristic scales of turbulence, the multiphase flow characteristics can be identified based on the Stokes number and volumetric fraction as well.

According to Figure (2.1), if the volume fraction is below $10^{-6}$, the flow can be treated as one-way coupled regardless of the Stokes’ number of particles. For the volume fraction values between $10^{-6}$ and $10^{-3}$ and the low Stokes’ number values, the flow can be considered as two-way
coupled. For the volume fraction values higher than $10^{-3}$, the flow can be treated as four-way coupled.

In two or four-way coupled flows, the investigation of turbulence modulation becomes important. Turbulence modulation refers the situation that the presence of a dispersed phase has an impact on the turbulence intensity, thus the eddy length, of the carrier flow. The presence of a dispersed phase may either augment or diminish the turbulence intensity of the multiphase flow compared to a single-phase flow. The turbulence intensity can be affected by particle size, volume fraction, characteristic length of flow, etc. According to Gore R. A. and Crowe C. T. (1989) [8], small particles have effects on diminishing the turbulent intensity of the carrier flow, whilst large particles augment the turbulent intensity. However, there are other models in the literature that associate turbulent modulation with either volume fraction or Reynolds number of the flow or other flow characteristics [7].

In the determination of flow type, one could also look at the coupling parameter besides Figure (2.1) proposed by Elghobashi (2006) [6] to investigate if the particle flow and the carrier flow are coupled. The mass, momentum, and energy coupling parameters can be defined to determine if the change of particle mass considerably affects the bulk flow if the drag force considerably affects the bulk flow, and if the heat transfer could significantly change carrier flow temperature, respectively.

Mass coupling exists when at least one of the components of multiphase flow changes its phase. The mass coupling parameter was given by Michaelides E.E., Sommerfeld M., and van Wachem B. (2023) as below [5].

$$\Pi_{mass} = \frac{L \dot{m} \bar{\rho}_d}{u \bar{m} \bar{\rho}_c}$$

(2.23)

In which $\dot{m}$ denotes phase change rate, for instance, evaporation rate, L denotes the length in parallel to the streamline along which the phase change occurs, u denotes the gaseous phase velocity, $\bar{m}$ denotes the total mass of the particles, $\bar{\rho}_d$ and $\bar{\rho}_c$ denotes the bulk density of the particles and the carrier phase, respectively. If it is found that the $\Pi_{mass} \ll 1$ the multiphase flow is not coupled in terms of mass and mass coupling could be neglected. For the cases where phase change does not occur since $\Pi_{mass} = 0$ the flow system is regarded as uncoupled in terms of mass.
Likewise, other coupling parameters can be defined. Michaelides E.E. et al. (2023) gave a momentum coupling parameter based on the drag force acting on the carrier flow \[5\]. A drag force formulation for Stokes flow, which refers to the flow such that the Stokes number is less than unity and refers to a case where particles smoothly follow the streamlines of the carrier flow, was given in Equation (2.14). This is the force that particles experience due to the drag. This is also the same force that the carrier flow experiences due to the presence of particles. Hence, by taking the ratio of momentum change due to the drag force to the momentum change of the flow, below expression is given,

\[
\Pi_{mom} = \frac{3\pi n L \mu_c D}{\alpha_C \rho_C u} \left(1 - \frac{v}{u}\right)
\] (2.24)

In which  \( n \) denotes the particle concentration, \( L \) denotes the length in parallel to the streamline along which the phase change occurs, \( \mu_c \) is the carrier phase dynamic viscosity, \( D \) is the particle diameter, \( \alpha_C \) denotes the volume fraction of the carrier flow, \( \rho_C \) is the material density of the carrier flow, \( u \) denotes the gaseous phase velocity, and \( v \) is the particle velocity. If it is found that the \( \Pi_{momentum} \ll 1 \) the flow is not momentum-coupled. Therefore, it can be considered that the velocity of the carrier flow does not significantly change while particles are being injected if \( \Pi_{mom} \ll 1 \).

One should also check if the mass flow system is coupled in terms of energy transfer between phases. Energy coupling can be investigated by checking whether the heat transfer between phases affects the temperature of the carrier phase significantly. The energy coupling parameter is given the expression below by Michaelides E.E. et al. \[5\],

\[
\Pi_{en} = \frac{\bar{\rho}_d}{\bar{\rho}_c} \left(\frac{12 k_c}{c_d D^2 \rho_d} \frac{L}{u} \right) \left(\frac{T_d - T_c}{T_c}\right)
\] (2.25)

In the above expression, \( \bar{\rho}_d \) and \( \bar{\rho}_c \) denote the bulk density of the dispersed and carrier phase respectively, \( k_c \) denotes the thermal conductivity of the carrier phase, \( c_d \) denotes the specific heat constant of the dispersed phase, \( D \) denotes the mean diameter of the dispersed particles, \( \rho_d \) is the material density of the dispersed particles, \( L \) is the length in parallel to the streamwise direction along which the heat transfer occurs, \( u \) is the velocity of the carrier phase, \( T_d \) and \( T_c \) denote the initial temperature values of the dispersed and carrier phase, respectively. If \( \Pi_{en} \) is found to be less than unity, the energy coupling between phases could be neglected.
It is stated that the order of magnitude of $\Pi_{en}$ is the same as the order of magnitude of $\Pi_{mom}$. Hence, the justification of one-way coupling for momentum transfer justifies the one-way coupling for energy transfer [5]. Based on these parameters, we will show that the sand particle-laden airflow that we are interested in is one-way coupled in Chapter (4.4).
3. Theory

In Chapter 3, we will first focus on the governing equations for a single-phase flow, and then we will introduce governing equations for multiphase flow. The governing equations of flow originated from the conservation of mass, the 1\textsuperscript{st} law of thermodynamics, and Newton’s 2\textsuperscript{nd} law. Therefore, to describe the flow of a fluid, we should mathematically express the flow based on conservation laws and Newton’s 2\textsuperscript{nd} law.

The governing equations of flow can be derived, by considering either fixed or moving control volume in space or considering a macroscale or an infinitesimally small control volume. The term control volume refers to a selected region in space, which can either be stationary or moving with the flow, which allows mass and energy transfer through the control surface. The decision to choose a different control volume will change the form of governing equations. Although the set of equations we are solving does not make any difference in obtaining the solution, different control volume considerations require us to choose different discretization schemes to solve the governing equations numerically.

3.1 Governing Equations of Single-phase Flow

We could handle the flow of fluid problems with different approaches. Those different approaches could be preferred in terms of achieving different goals. Some of the approaches that we could use to handle fluid flow are summarized below:

- **Fixed Infinitesimal Element**: In this approach, the infinitesimally small, fixed object is considered, and the conservation equation is applied to this element. When we apply conservation equations, we obtain governing equations in differential form such that all the flow properties appear in differential form. This form is called the strong conservation form of the governing equations. In addition, the consideration of fixed control volume is called the Eulerian approach. By applying the Eulerian approach to infinitesimally small elements, we can get the below differential equation for mass conservation,

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho v)}{\partial t} + \frac{\partial (\rho w)}{\partial t} = 0
\]  

(3.1)
• **Moving Infinitesimal Element:** In this approach, the infinitesimally small, moving object is considered, and the conservation equation is applied to this element. In this consideration of control volume, we obtain governing equations in the form of differential equations; however, some flow properties appear outside the differential terms. Therefore, the set of equations that we obtained is called a strong, non-conservation form. In addition, the consideration of moving control volume is called the Lagrangian approach. We can get the below differential equation for mass conservation,

\[ \frac{\partial \rho}{\partial t} + u \frac{\partial (\rho)}{\partial x} + v \frac{\partial (\rho)}{\partial y} + w \frac{\partial (\rho)}{\partial z} + \rho \nabla \mathbf{V} = 0 \]  

(3.2)

• **Fixed Finite Element:** In this approach, the finitely small, moving object is considered, and the conservation equation is applied to this element. In this consideration of control volume, we obtain governing equations in integral form in which all flow properties appear under differentials. Therefore, the set of equations that we obtained is called the weak conservation form. The weak form of the equations allows discontinuities in the flow properties, unlike the strong form of the equations mentioned above. The mass conservation equation for this approach can be given below,

\[ \frac{\partial}{\partial t} \iiint_V \rho dV + \oint_S \rho \mathbf{V} dS = 0 \]  

(3.3)

• **Moving Finite Element:** If we consider moving finite elements, we obtain weak non-conservation forms of equations. As mentioned above, the weak form of the equations refers to the equations in which all terms appear in integral form. Conservation form refers to the case where all terms appear inside of the derivative, and non-conservation form is obtained if at least one term is outside of the derivative.
Hence, we can write below the mass conservation equation for weak non-conservation form,

$$\frac{\partial}{\partial t} \iiint_V \rho dV + \mathbf{u} \frac{\partial}{\partial x} \iiint_V \rho dV + \mathbf{v} \frac{\partial}{\partial y} \iiint_V \rho dV + \mathbf{w} \frac{\partial}{\partial z} \iiint_V \rho dV = 0 \quad (3.4)$$

In the above equations, \( \rho \) denotes the density of the fluid \( u, v, \) and \( w \) denotes the velocity components of the flow in \( x, y, \) and \( z \) directions respectively, \( V \) denotes the control volume element, and \( S \) denotes the control surface, \( \mathbf{V} \) denotes the velocity of flow.

In this thesis, we are going to solve a multiphase flow whose components are air as a carrier medium and sand as discrete particles by using an OpenFOAM solver. OpenFOAM (Open Field Operation and Manipulation) is a toolbox written in a C++ environment that solves governing equations of flow by considering finite control volume. Therefore, the equations that we are solving appear in the weak, i.e., integral form. Depending on our interest in treating flow from Eulerian or Lagrangian perspectives, we could either deal with the conservation or non-conservation form of the equation.

By applying conservation laws, the continuity, Navier-Stokes, and energy equations can be written in strong conservation for a single-phase flow as below:

- **Continuity Equation**

  $$\frac{\partial \rho}{\partial t} + \nabla (\rho \mathbf{V}) = 0 \quad (3.5)$$

- **Navier-Stokes Equations**

  $$\frac{\partial (\rho u)}{\partial t} + \nabla (\rho u \mathbf{V}) = - \frac{\partial \rho}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + \rho f_x \quad (3.6)$$

  $$\frac{\partial (\rho v)}{\partial t} + \nabla (\rho v \mathbf{V}) = - \frac{\partial \rho}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} + \rho f_y \quad (3.7)$$
\[
\frac{\partial (\rho w)}{\partial t} + \nabla \cdot (\rho w V) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho f_z
\]  

Energy equation

\[
\frac{\partial}{\partial t} \left( \rho \left( e + \frac{V^2}{2} \right) \right) + \nabla \cdot \left( \rho \left( e + \frac{V^2}{2} \right) V \right) = \rho q_{rad} + \nabla \cdot q_{cond} - \frac{\partial (\rho u p)}{\partial x} - \frac{\partial (\rho v p)}{\partial y} - \frac{\partial (\rho w p)}{\partial z} = \nabla \cdot (\tau V) + \rho f \cdot V
\]

For the above equations, again \( \rho \) denotes the density of the fluid, \( u, v, w \) denotes the velocity components of the flow in \( x, y, z \) directions respectively, \( V \) denotes the velocity vector of the flow, \( p \) denotes the pressure, \( V \) denotes the magnitude of the velocity, \( \tau \) denotes the stress tensor, \( q_{rad} \) denotes the heat transfer due to radiative heating, \( q_{cond} \) denotes the heat transfer due to convection, \( f \) denotes the body forces per unit mass that act on the fluid.

Conductive heating can be expressed as below,

\[ q_{cond} = -k \nabla T \]  

In which \( k \) is the thermal conductivity of the fluid, which depends on the temperature, pressure, and the composition of flow.

Apart from the above governing equations, the chemical species conservation equation should also be considered to include the effects of possible chemical reactions. According to Fatti V. and Fois L. (2021) [10], the chemical species conservation equation can be expressed as below,

\[ \frac{\partial \rho_i}{\partial t} + \nabla \cdot (\rho_i V) = \nabla \cdot j_i + R_i \]  

In which \( \rho_i \) is the material density of \( i^{th} \) component, \( j_i \) is the diffusive flux of the \( i^{th} \) component, and \( R_i \) is the reaction term. The diffusion flux can be rewritten through the usage of Fick’s law as below,

\[ j_i = -\Gamma_{i,\text{mix}} \nabla Y_i \]  

(3.12)
Where \( \Gamma_{i,mix} \) denotes the mass diffusion coefficient, \( i^{th} \) species with respect to the mixture and \( Y_i \) is the mass fraction of the \( i^{th} \) component of the flow. Therefore, \( \nabla Y_i \) denotes the mass fraction gradient of that component.

### 3.2 Governing Equations of Multiphase Flow

While we are deriving the governing equations of multiphase flow, apart from the fact that we could consider either the weak or the strong form of the equations, we could also treat the dispersed phase in different ways. The specific treatment of the dispersed phase will modify the governing equations of the multiphase flow and yield us the flexibility of choosing either obtaining a high resolution of the motion of the particles that constitute the dispersed phase or obtaining a solution in such a way that we could track the bulk amount of flow instead of tracking particles.

Although there are other approaches, we will discuss below two approaches to handle multiphase flow:

- Eulerian-Eulerian Approach
- Eulerian-Lagrangian Approach

**Eulerian-Eulerian Approach:** In this approach, both the carrier medium and dispersed phases are considered as a continuous medium, so that the control volume contains a sufficiently large number of particles. In this case, the continuity equation, momentum equations, and the conservation of energy equation can be applied to both carrier medium and dispersed phase, and those equations can be solved simultaneously by considering phase changes and fluid-particle couplings.

However, in this approach, since we consider both phases continuous, Eulerian-Eulerian multiphase flow modeling is reported to fail when the volume fraction of the dispersed phase becomes too low, as the particulate flow can no longer be considered continuous [11].

**Eulerian-Lagrangian Approach:** In this approach, the carrier medium is considered continuous, whilst the discrete particles are tracked individually. The Eulerian-Lagrangian model has the advantage of tracking particles so that we can observe particle-wall collisions and
particle-particle collisions in contrast to the Eulerian-Eulerian frame. In a one-way coupled flow case, we could first solve the carrier flow in the Eulerian frame, and then we can evolve the particle as a post-processing transaction by considering this flow since the carrier flow is independent of the particle that the fluid contains in the one-way coupled system. This is known as Lagrangian particle tracking.

In this Eulerian-Lagrangian framework, firstly, the Eulerian equations of carrier medium flow are solved, and then the flow solution of carrier medium is interpolated at the grid nodes to the particle position in order to determine the forces acting on each particle. Afterward, the particle trajectories can be drawn by applying Newton’s second law to the particles. Another critical feature of the Eulerian-Lagrangian framework is the need to choose a proper time step to capture particle trajectories with sufficiently high resolution so that one cannot lose the information of particle-wall collision or other significant interactions that are happening in multiphase flow. On the other hand, selecting a time step too low would cause an increment in the computational cost. Hence, selecting a proper time step and the interpolation of the Eulerian solution of the carrier medium at particle positions are of great importance.

### 3.2.1 Governing Equations for Eulerian-Eulerian Approach

In the Eulerian-Eulerian approach, we treat both the discrete phase and the carrier medium as continuous medium. In this consideration of the multiphase flow, we treat both phases as fluid. Hence, this consideration will allow us to apply continuity, Navier-Stokes, and energy equations discussed in Chapter 3.1. This approach would allow us to reduce the computational cost since we do not need to track dispersed phase particles, but rather, we could treat dispersed phase particles as fluid by applying some source terms to consider inter-particle interactions of the dispersed phase. The continuity equation for this approach can be given as below,

\[
\frac{\partial}{\partial t} (\alpha_\psi \rho_\psi) + \nabla \cdot (\alpha_\psi \rho_\psi \mathbf{U}_\psi) = 0
\]  

(3.13)

In the above equation, \( \alpha, \rho, \) and \( \mathbf{U} \) represent the volumetric fraction, material density, and bulk velocity, respectively. The subscript \( \psi \) indicates the phase we are interested in; therefore, Equation (3.13) should be solved for both the dispersed phase and the carrier medium.
For the momentum, we could write the below equations for the carrier medium and the dispersed phase respectively.

\[
\frac{\partial}{\partial t} (\alpha_c \rho_c \mathbf{U}_c) + \nabla \cdot (\alpha_c \rho_c \mathbf{U}_c \mathbf{U}_c) = \nabla \cdot \tau_c - \alpha_c \nabla p + \alpha_c \rho_c \mathbf{g} - \mathbf{F}_D \tag{3.14}
\]

\[
\frac{\partial}{\partial t} (\alpha_d \rho_d \mathbf{U}_d) + \nabla \cdot (\alpha_d \rho_d \mathbf{U}_d \mathbf{U}_d) = \nabla \cdot \tau_d - \alpha_d \nabla p + \nabla P_s + \alpha_d \rho_d \mathbf{g} + \mathbf{F}_D \tag{3.15}
\]

\[
\mathbf{F}_D = \frac{1}{2} C_d A_{cs} \rho_d |U_c - U_d| (U_c - U_d) \tag{3.16}
\]

In the above equations, \(\alpha, \rho,\) and \(\mathbf{U}\) again represent the volumetric fraction, material density and bulk velocity, respectively. The subscripts c and d denote carrier and discrete phase properties, respectively. \(\tau\) denotes phase stress tensor, \(p\) denotes pressure field, \(\mathbf{g}\) denotes gravity vector, and \(\mathbf{F}_D\) denotes the drag force.

As per Newton’s 3\(^{rd}\) law, in Equation (3.14) and Equation (3.15), we used opposite signs while expressing the drag forces both for the carrier medium and the dispersed phase since the drag has an effect of decelerating the carrier flow and accelerating the particles when carrier medium velocity is higher than the velocity of dispersed particles. In Equation (3.15), in addition to the pressure gradient, we consider the particle pressure gradient term, which is denoted by \(\nabla P_s\) according to the model proposed by Gidaspow D., (1994) \cite{9}.

In Equation (3.16), the drag force that we considered was given. In this equation, \(C_d\) denotes the drag coefficient, \(A_{cs}\) denotes the particles’ cross-sectional area, and the drag coefficient is generally described as a function of relative Reynolds number, which will be further elaborated in Chapter (3.3.1).

For the conservation of energy, we could write the below equation for a two-component system by including energy exchanges due to chemical reactions,

\[
\frac{\partial}{\partial t} (\alpha_{\psi} \rho_{\psi} (H_{\psi} + K_{\psi})) + \nabla \cdot (\alpha_{\psi} \rho_{\psi} \mathbf{U}_{\psi} (H_{\psi} + K_{\psi})) - \nabla \cdot ((\alpha_{\psi} \alpha_{\psi}^{\text{eff}} \nabla H_{\psi})) =
\]

\[
= \alpha_{\psi} \frac{\partial p}{\partial t} + \rho_{\psi} \mathbf{g} \cdot \mathbf{U}_{\psi} + \alpha_{\psi} \rho^{r}_{K} + \rho^{r}_{E} + \rho^{r}_{T}
\]

\[
\tag{3.17}
\]
In which $H_\psi$ is the enthalpy of the phase, $K_\psi$ is the kinetic energy of the phase, $\alpha_\psi^{\text{eff}}$ denotes the effective thermal diffusivity, $Q_R^\cdot$ denotes energy exchange between phases due to chemical reactions, $Q_{KE}^\cdot$ is the energy exchange between phases due to kinetic energy differences between phases, and $Q_{TD}^\cdot$ is the energy exchange between phases due to the temperature differences between phases [10].

### 3.2.2 Governing Equations for Eulerian-Lagrangian Approach

In this approach, we model carrier medium from the standpoint of the Eulerian framework, while we model the dispersed phase from the perspective of the Lagrangian framework.

This approach applies the continuity, Navier-Stokes, and energy conservation equations to the carrier medium. In contrast, we apply Newton’s second law to the dispersed particles to estimate particle trajectories.

For the carrier medium, the continuity, Navier-Stokes, and energy conservation equations can be written by considering that the only fluid material is the carrier medium. Hence, by removing the volumetric fraction term from Equations (3.13), (3.14), and (3.17) and by substituting the subscript $c$ into subscript $\psi$, we can obtain the governing equations of the carrier medium.

For the dispersed phase, according to Newton’s second law, the below equations can be written,

$$m_i \frac{\partial U_i}{\partial t} = \sum_{j=1}^n n_i^c F_{i,j}^c + \sum_{k=1}^n n_{nc}^i F_{i,k}^{nc} + F_{i}^f + F_{i}^g$$  \hspace{1cm} (3.18)

$$I_i \frac{\partial \omega_i}{\partial t} = \sum_{j=1}^n n_i^c M_{i,j}$$  \hspace{1cm} (3.19)

$$\frac{\partial x_p}{\partial t} = U_p$$  \hspace{1cm} (3.20)

In Equation (3.18), $U_i$ is the velocity of the particle $i$, $m_i$ is the mass of particle $i$, $F_{i,j}^c$ denotes the contact force acting on particle $i$ by particle $j$ or wall, $F_{i,k}^{nc}$ represents non-contact force acting on particle $i$ due to the particle $k$, $F_{i}^f$ denotes the force arising from the particle-fluid interactions, and $F_{i}^g$ is the gravitational force acting on particle $i$. Moreover, in the sum
indices, \( n^c_i \) and \( n^{nc}_i \) denote number of total contacts for particle i and number of total non-contact for particle i.

In Equation (3.19), \( I_i \) denotes the moment of inertia of the particle i, \( \omega_i \) denotes the angular velocity of particle i, and \( M_{i,j} \) denotes the contact moment acting on particle i due to particle j.

In equation (3.20), \( x_p \) denotes the position vector of particle i, and \( U_p^i \) denotes the velocity vector of particle i.

The Eulerian-Lagrangian approach would allow us to track particles more easily than the Eulerian-Eulerian approach since we have dynamic equations of particles, i.e., Equations (3.18) to (3.20). This approach could be adapted to track particles in the multiphase flow where the dispersed phase fraction is too small to treat flow by using the Eulerian-Eulerian approach. As the number of particles in the dispersed phase increases, tracking all particles will become computationally costly.

### 3.3 Forces Acting on a Single Particle

In this section, we will discuss some of the forces acting on the particles to show which forces on the right-hand side of the Equation (3.18) can be considered. In this thesis, we considered the only drag, buoyancy, and gravitational force acting on a single particle. However, several other sources act or induce force on a particle, such as transverse lift forces, electrical forces, forces due to Brownian motion, and Basset force that occurs due to transient flow conditions etc.

#### 3.3.1 Drag Force

When the relative velocity between the carrier medium and the dispersed phase is constant, the carrier medium exerts a steady drag force on dispersed particles. In general, this drag force is given as a function of the relative Reynolds number of the particles, which can be given as below expression,

\[
Re_r = \frac{D_p |u-v| \rho_c}{\mu_c}
\]  

(3.21)
In this equation, $D_p$ denotes the diameter of the particle, $u$ denotes the carrier phase velocity, $v$ denotes the dispersed particle velocity, $\rho_c$ and $\mu_c$ denote the carrier phase's material density and the carrier phase's dynamic viscosity, respectively.

Furthermore, we could express the steady drag force acting on a particle based on dimensionless drag coefficient $C_d$, as below,

$$ F_d = \frac{1}{2} \cdot C_d \cdot A_{cs} \cdot \rho_d \cdot |u - v| \cdot (u - v) $$

(3.22)

In most cases, the drag coefficient can be expressed as a function of relative Reynolds number. It is stated by Michaelides E. E. et al. (2023), that for very low Reynolds number which is also called Stokesian or creeping flow, the effects of fluid inertia on the particle diminish, and a fore and aft symmetry occurs in the flow around the spherical particles [5]. For this case, Hadamard, J.S. (1911) [12] and Rybczynski, W. (1911) [13] independently obtained an analytical expression for the drag force and therefore for the drag coefficient for the Stokesian flow. The drag coefficient for the Stokesian flow is given below,

$$ C_d = \frac{8(3\lambda+2)}{Re_r(\lambda+1)} $$

(3.23)

In Equation (3.23) $\lambda$ denotes ratio of dynamic viscosity of dispersed phase to dynamic viscosity of carrier medium, i.e $\lambda = \frac{\mu_d}{\mu_c}$.

If the dispersed phase is formed by solid particles, the viscosity ratio approaches infinity, so that we could find drag coefficient for solid particles by taking the limit of the equation (3.23). Hence, for solid particles we find below drag coefficient expression,

$$ C_d = \frac{24}{Re_r} $$

(3.24)

The drag coefficient which is given by Equation (3.24) is only valid for Stokesian flow in which the relative Reynolds number of the particles very small, i.e $Re_r \ll 1$.

When the relative Reynolds number of particles increases, the fore-aft symmetry around the particles deteriorates and a flow wake downstream of the particle occurs. Based on the reference Ph.D. thesis written by Kristopher T. Olshefski (2023) [1], we found that the relative
Reynolds number of particles is around 689 therefore, we cannot treat this flow as Stokesian flow.

For the high relative Reynolds number, a wake downstream of the spherical particles occurs and the drag coefficient differs from the Stokesian flow by a drag factor.

For this high relative Reynolds number flow, there are some empirical and semi-empirical expressions that can be used for calculating the drag coefficient. One of them is developed by Schiller L. and Naumann A. (1933), which can be given as [14],

\[
\begin{align*}
\{C_d &= \frac{24}{Re_r}(1 + 0.15Re_r^{0.687}) \quad Re_r \leq 1000 \\
\{C_d &= 0.44 \quad Re_r > 1000 \}
\end{align*}
\] (3.25a)

Another expression that we will use in our simulation was given by Putnam A. (1961) is expressed as below [14],

\[
\begin{align*}
\{C_d &= \frac{24}{Re_r}(1 + \frac{1}{6}Re_r^{2}) \quad Re_r \leq 1000 \\
\{C_d &= 0.424 \quad Re_r > 1000 \}
\end{align*}
\] (3.26a)

3.3.2 Body Forces

The gravitational force acts on every particle which can be expressed by including the buoyant force as below,

\[F_G = \frac{\pi D_v^3}{6} (\rho_d - \rho_c) g \] (3.27)

In which \(D_v\) denotes the volume equivalent of the particle diameter, \(\rho_d\) and \(\rho_c\) denote material density of the dispersed and carrier phase, respectively, and \(g\) denotes the gravity vector.

3.3.3 Other Forces

There are also some other forces acting on particles, which may be resulted from electric and magnetic fields, transverse lift forces due to the rotation of particles or the Brownian motion of the particles. We will briefly look at the transverse lift forces.
One of the transverse lift forces that may act on the particle is called Magnus force proposed by Magnus G. (1861). When the particle moves and rotates with respect to the far-field flow, a force proportional to the particle relative velocity and particle angular velocity act on the particle, in the direction of which perpendicular to the plane formed by particle relative velocity and its angular velocity axis. This force is given as below [5],

\[
F_{LM} = \frac{\pi}{8} D^3 \rho_c \Omega x (u - v)
\]  

(3.28)

In which \(F_{LM}\) denotes Magnus lift force, \(D\) is the diameter of the particle, \(\rho_c\) denotes the material density of the carrier medium, \(\Omega\) is angular velocity vector, \(u, v\) denote the carrier medium velocity and the dispersed particle velocity, respectively.

Another transverse lift force that may act on a particle is called Saffman lift force proposed by Saffman P.G. (1968). This force stems from the shear flow where a velocity gradient in the carrier medium flow occurs in the direction perpendicular to the particle velocity. This force can be expressed as below [5],

\[
F_{LS} = \frac{1.615D^2 \sqrt{\rho_c \mu_c}}{\sqrt{||\gamma||}} (u - v) x \gamma
\]  

(3.29)

In which \(F_{LS}\) denotes Saffman lift force, \(D\) is the diameter of the particle, \(\rho_c\) and \(\mu_c\) denotes the material density of the carrier medium and the dynamic viscosity of the carrier medium respectively, \(\gamma\) is the shear velocity vector, \(u, v\) denote the carrier medium velocity and the dispersed particle velocity, respectively.

Apart from these forces, other forces related to Brownian motion, or electro-magnetic forces can act on particles. However, for the scope of this master thesis, we will only consider the presence of drag force in which drag coefficient was given by Putnam A. (1961), and the buoyancy force.

In addition to the forces acting on particles, other forces may arise due to particle-fluid interaction and particle-particle collision. However, our interest in this master’s thesis is to numerically investigate a dilute flow, which will be shown that the multiphase flow that we are interested in is one-way coupled flow in Chapter (4.4), in which we can neglect dispersed phase flow effect on the carrier medium and particle-particle collisions. Therefore, we will not
elaborate on dispersed phase flow effect on the carrier medium and particle-particle collisions; further information can be found in the book written by Michaelides, E., E. et al. (2023) [5].

3.4 Erosion Models

Erosion is once defined as “material damage caused by the attack of particles entrained in a fluid system impacting the surface at high speed,” Bitter J.G.A, (1963a) [5], and in this thesis, we will investigate the erosion of blade materials by using some empirical approaches. Erosion can be defined by using different units, which might be given in terms of (mm/kg), which is depth of erosion per unit kilogram of impinging particles, or it can be given as (kg/m^2s) which is weight of eroded material per unit area and the erosion time. Erosion can also be given as a dimensionless unit (kg/kg), which is the weight of material loss per unit impinging particles.

There are several erosion models in the literature that estimate the erosion either in terms of erosion rate, or the eroded material weight. In this thesis, we will use the erosion model which was developed by Finnie I., (1960) [15] to estimate the eroded volume of compressor blade per unit span of the compressor blade in the units of (mm^3/m). In the following subsections, we will discuss some of the erosion models.

3.4.1 Finnie Model

The Finnie model was proposed by Finnie I., (1960) to estimate the eroded volume of material in (m^3) as a function impinging particle mass and particle velocities. It is stated that the model was developed for ductile materials. The Finnie model equations can be expressed as below [15],

\[
E = \frac{M u_p^2}{p \psi K} \left( \sin(2\alpha) - \frac{6}{K} \sin^2(\alpha) \right) \quad \text{for } \alpha \leq \frac{6}{K} \quad (3.30a)
\]

\[
E = \frac{M u_p^2}{p \psi K} \left( \frac{K \cos^2(\alpha)}{6} \right) \quad \text{for } \alpha > \frac{6}{K} \quad (3.30b)
\]

In the Equations (3.30a) and (3.30b) \( \alpha \) denotes the impingement angle, \( M \) denotes the total mass of the particles, \( u_p \) is the particle impact velocity, \( \psi \) denotes the ratio of depth of contact to depth of cut, \( K \) denotes the ratio of normal force to tangential force, \( p \) denotes the flow stress of eroded material. In this expression, \( K \) and \( \psi \) are both assumed to be 2 by Finnie I.,
(1960), and it is stated that flow stress can be considered as 2.7 GPa, in the book written by Michaelides E. E., et al. (2023) [5].

We will use the Finnie model and the above given values while calculating the eroded volume in this thesis.

3.4.2 Neilson and Gilchrist Model

Another erosion model that can be used for estimating the eroded weight was proposed by Neilson, J.H. and Gilchrist, A. (1968). This model estimates the erosion by considering both the cutting wear and deformation wear. Therefore, it can be expressed as,

\[ E = E_C + E_D \] (3.31)

In which \( E_C \) and \( E_D \) denote the eroded weight due to cutting wear and deformation wear, respectively. \( E_C \) can be given as [16],

\[ E_C = \frac{M u_p^2 (\cos^2(\alpha)) \sin(n \alpha)}{2 \phi_c} \] for \( \alpha < \alpha_0 \) (3.32a)

\[ E_C = \frac{M u_p^2 \cos^2(\alpha)}{2 \phi_c} \] for \( \alpha > \alpha_0 \) (3.32b)

\[ n = \pi / 2 \alpha_0 \] (3.33)

In the Equations (3.32a) and (3.32b) \( \phi_c \) denotes the required kinetic energy to erode one unit mass of the material that is being eroded, \( \alpha_0 \) denotes the transition angle. For the estimation of erosion due to deformation, below expression is given by Neilson, J.H. and Gilchrist, A. (1968),

\[ E_D = \frac{M (u_p \sin(\alpha) - K)^2}{2 \epsilon_c} \] (3.34)

In the Equation (3.34), \( K \) denotes the threshold normal velocity component below which erosion due to deformation is neglected and \( \epsilon_c \) denotes the deformation coefficient which is the required kinetic energy to erode one unit mass. According to Michaelides E. E., et al. (2023), \( \phi_c \) is equal to 3.332x10\(^7\) and, \( \epsilon_c \) is equal to 7.742x10\(^7\) [5].
3.4.3 Zhang Model

The Zhang model was proposed by Zhang Y., et al. (2007), which estimates the erosion as a rate, in terms of the ratio of the mass of the loss of material to the mass of the impinging particles, as per following empirical equation,

\[ E = C (BH)^{-0.59} F_s V_p^n F(\alpha) \]  (3.35)

where \( C \), and \( n \) are empirical constants, \( BH \) denotes the Brinell hardness value of the eroded material, \( F_s \) denotes shape factor, \( V_p \) is the velocity of the particles, and \( F \) is a function of impact angle \( \alpha \), such that \( F(\alpha) \) is given as below,

\[ F(\alpha) = 5.4\alpha - 10.11\alpha^2 + 10.93\alpha^3 - 6.33\alpha^4 + 1.42\alpha^5 \]  (3.36)

\( C \) and \( n \) are assumed to be equal to \( 2.17 \times 10^{-7} \) and \( 2.41 \), respectively, \( F_s \) is the shape factor, which is 0.2 for spherical particles, 0.53 for semi-rounded particles and 1 for sharp particles [5].
4. Experimental Setup

The initial and boundary conditions for this numerical investigation research of the effects of sand particle ingestion on compressor blade erosion have been obtained from the work done by Kristopher T. Olshefski in the Advanced Power and Propulsion Laboratory (APPL) at Virginia Tech under the supervision of Dr. K. Todd Lowe and Dr. Wing F. Ng. in 2023. In the dissertation thesis of Kristopher T. Olshefski, the process of developing two different diagnostic tools and their implementation to the turbo-shaft engine system has been explained in the hope of having a better scientific understanding of the multiphase flow and sand ingestion phenomena.

In this section, we will first discuss the turboshaft engine test cell located in APPL at Virginia Tech. Secondly, we will provide the information on diagnostic tools that have been employed to make measurements related to the determination of the sand particle volume fraction and its variation. In the third stage, we will share some results of the thesis written by Olshefski, K. T., and we will infer initial and boundary conditions such as the volumetric fraction of sand particles and the chord length of the compressor blade. From this inferral, we will estimate the initial boundary conditions and we will solve the multiphase problem to estimate the erosion of compressor blade due to sand particles impingement.

4.1 Test Cell

The test cell was developed to conduct experiments to investigate the effects of sand ingestion on turboshaft engines as a consequence of research endeavors over several years. The test cell mainly includes the Rolls-Royce M250-C20B turboshaft engine, water brake dynamometer, oil system, fuel system, water system, instrumentation, and sand delivery system.

The engine that was used for conducting sand ingestion experiments in APPL is a Rolls-Royce M250-C20B turboshaft engine, whose take-off power equals to 420 SHP. The engine consists of 6 stages axial compressor and a single-stage centrifugal compressor, two gas generator turbine stages that drive the compressor, and two power turbine stages that are connected to the Kahn water brake dynamometer [1].

The water brake in the abovementioned scheme is used for loading the engine, and it is used for absorbing mechanical energy by converting mechanical energy into the increase in the
temperature of water that drives the water brake. To supply water to the water brake dynamometer, the test cell had a water system containing a cold-water tank, water pumps, control valves, an intermediate water tank, and an oil-water heat exchanger. Apart from supplying the required water for the brake, the water system is also used for oil cooling.

The oil system is utilized to supply steady oil for the lubrication purposes of the engine components. The oil system is comprised of an oil tank, oil filter, heat exchanger to limit the oil temperature to 200 °F, and several pressure transducers and thermocouples.

The fuel system is also utilized for steady fuel supply, which is comprised of a fuel tank whose capacity is 180 gal, a fuel pump, a pressure regulator to ensure that the pressure of the fuel is approximately 25 psi, a fuel filter, solenoid valves and several measurement instruments such as flowmeter, thermocouples, etc. Solenoid valves are utilized in the fuel system to ensure fuel flow is turned off in an emergency.

The engine instrumentation system comprises required instruments to measure inlet total temperature and pressure (mass flow measurement), engine core speed, output shaft speed, torque, interstage turbine temperature, and fuel mass flow rate.

The sand delivery system is utilized to supply sand particles steadily to experimentally simulate the sand ingestion phenomena that an aircraft may encounter under dusty conditions. In the reference Ph.D. thesis, it is stated that “The sand delivery system was designed at Virginia Tech and utilized in the particle ingestion research consists of four main elements – a sand hopper mounted to a scale inside an enclosure, tubing with a vacuum conveyor, a delivery nozzle, and a flow barrel with a flow conditioner.” [1]. Sand particles are taken from the sand hopper with the aid of a helical screw element, which is supplying sand particles by rotating its longitudinal axis. With the capacity to change the screw element rotational speed remotely, the calibration of the sand particle supply rate becomes possible. A vacuum conveyor, another system's main component, is utilized to ensure that the particles can reach the delivery nozzle inside the flow barrel.
4.2 Anisokinetic Probe

The diagnostic tools that have been developed as a part of the above-mentioned project [1] include an anisokinetic sand probe. It is claimed that the usage of this probe would be helpful to access real-time sand ingestion data during the operation of an aircraft. An isokinetic sand probe can be used for measuring local mass flux, and bulk density of flow under certain conditions.

The term isokinetic refers to an ideal condition where the probe is able to draw sand particles proportional to the probe cross-sectional area. This condition can be met when the local flow velocity at which the sand probe is located can be equalized to the suction velocity. If we define the area in which the sand particles are drawn into the sand probe as the capture area, the isokinetic condition refers to the condition where the capture area of the sand particle is equal to the sand probe cross-sectional area.

When the suction velocity inside the sand probe becomes higher than the local velocity of the flow, the super isokinetic sampling condition occurs. In this case, the captured area becomes greater than the sand probe cross-sectional area. This situation leads to the oversampling of small Stokesian particles [1].

When the suction velocity inside the sand probe becomes lower than the local velocity of the flow, the sub-isokinetic sampling condition occurs. In this case, the captured area becomes smaller than the sand probe cross-sectional area. This situation leads to the under-sampling of small Stokesian particles. Whereas the particles that have higher Stokes numbers are found less sensitive to the sub-isokinetic or super-isokinetic conditions by Olshefski K. T. (2023) [1].

In addition to this, Bohnet M. (1973) developed a correlation for the ratio of the capture area to the probe cross-sectional area. Based on this correlation, he correlated his results with a non-dimensional Stokes number which is expressed as,

\[ \text{St}_{pr} = \frac{\rho_d D^2 u_0}{18 \mu_c D_{pr}} \]  \hspace{1cm} (4.1)

In which \( \text{St}_{pr} \) is non-dimensional Stokes number, \( \rho_d \) is the material density of the dispersed phase, \( D \) is particle diameter, \( u_0 \) is the local flow velocity, \( \mu_c \) dynamic viscosity of the
carrier medium, and $D_{pr}$ is probe diameter. In his work, Bohnet M. shows that the concentration of particles that are obtained under super-isokinetic conditions is less than the isokinetic conditions, and the concentration of particles that are obtained under sub-isokinetic conditions is higher than the isokinetic conditions. Bohnet M. also shows that the deviation from the isokinetic condition increases with increasing Stokes number [18].

In addition, the misalignment between the sand probe and the flow stream also causes anisokinetic sampling conditions, i.e. either sub-isokinetic or super-isokinetic conditions. Therefore, the isokinetic sampling condition delineates ideal conditions in terms of more precisely sampling the discrete phase of the multiphase flow.

4.3 An Estimation for the Chord Length of the Compressor Blade

In the reference Ph.D. thesis [1], it is stated that an anisokinetic probe was developed to perform an aerodynamic study of the sand probe, a particle tracking study, and a particle sampling study. The probe was tested for the Mach numbers of 0.25 and 0.70 and the yaw angles for the range of 0° to 45° degrees. The yaw angle term here represents the misalignment between the flow stream and the sand probe orientation.

In the reference Ph.D. thesis [1], it is stated that the Stokes number of particles for C-spec sand, whose mean diameters are 250 μm, varies between 850<St<1170 where the Mach number varies from 0.25<M<0.70. Regarding this information, a rough estimation for the chord length of the first stage of the compressor blade can be made as follows,

\[ St = \frac{ρ_d D^2 U}{18μc} \]  \hspace{1cm} (4.2)

\[ a = \sqrt{\frac{γR \bar{T}}{M}} \]  \hspace{1cm} (4.3)

In the above equations, c represents chord length, a is the speed of sound, $γ$ is the adiabatic index, $\bar{R}$ is the universal gas constant, and M is the mean molar mass of air. We know that the material density of sand particles is 2650 kg/m³, the mean diameter of sand particles is 250 μm, and the dynamic viscosity of air at 20 °C is 18.1 μPa.s. Therefore, by substituting the values into equations, we obtain that,
52 mm < chord length < 146 mm where \(0.25 < M < 0.70\) \& \(St=850\)

In this thesis, the chord length was initially selected as 63 mm, and then the above range has been found for the chord length. Since 63 mm falls into the above range, the chord length of 63 mm kept unchanged. Another study, conducted by Zhu W. et al. investigates the effect of tip clearance on the performance of an axial compressor rotor and showed the effect of tip clearance on a compressor blade whose chord length to tip clearance ratio is 450 \([19]\).

In the reference Ph.D. thesis, it is shown that the tip clearance of the first stage is about 5 mil \([1]\). If we use the given ratio above that corresponds to a chord length of 57 mm. Since the proximity of these values to the initially selected value of 63 mm, the chord length was kept unchanged for the convenience of geometry creation to create the numerical domain and considered as 63 mm.

4.4 Particle Visualization by Illuminated Scattering

A particular illumination technique which is called “ParVIS” stands for “Particle Visualization by Illuminated Scattering” was developed as a part of the reference Ph.D. thesis \([1]\). The results obtained by this diagnostic tool will include helpful information to set initial conditions for the volumetric fraction ratio of the dispersed particles. Before sharing the initial results, we would like to elaborate on the ParVIS diagnostic tool.

The ParVIS diagnostic tool contains a laser light source and a camera to allow users to obtain qualitative information about the volume fraction of particles. In the reference Ph.D. thesis, it is stated that the mass concentration of the discrete particles is proportional to the light intensity. The formulation that correlates the mass flow concentration of the discrete phase to the intensity of the light source can be found in the work of Yu X., Shi Y., et al (2017) \([17]\).

In the reference Ph.D. thesis, the mass flow concentration of sand particles is shared for both high-concentration and low-concentration cases. To obtain initial results for mass concentration, this image should be converted into matrix form. Since the light intensity is proportional to the mass concentration a calibration constant that turns the numbers in the matrix into mass concentration can be found. Afterward, we need to find the line where the sand particle concentration is high since we intend to solve the multiphase problem in 2D. Then, based on
mass concentration information, volumetric fraction and the number of particles that are introduced into the system per second can be found. On one hand, having the information of volumetric fraction is important for us to evaluate the flow in terms of fluid-particle coupling. On the other hand, knowing the number of injected particles per second is required for tracking particles accurately. Below results that show the mass concentration are taken from [1].

\[ \dot{M} = 1.68 \text{ g min}^{-1} \]

\[ \dot{M} = 0.55 \text{ g min}^{-1} \]

**Figure 4.1:** The mass concentration date of sand particles both for high loading, is on the right, and low loading conditions, is on the left. Taken from Olshefski K.T. (2023) [1], with author’s permission.

In the above graph, the high-loading case refers to a case where the sand particle concentration is 45 mg/m\(^3\) and the low-loading case refers to a case where the sand particle
concentration is $22.5 \text{ mg/m}^3$ [1]. While we are obtaining the initial conditions for volumetric fraction, we used the high-loading case whose concentration values are shown in Figure (4.1) on the left image. To do so, the image was converted into 16-bit images. The part of the image that contains only white color while importing the image into MATLAB was cut and we obtained a 560x160 matrix. This matrix is calibrated to show density distribution. Then, since we will be interested in multiphase flow in 2D, we need to find the line parallel to the $y$-axis where the highest bulk density of sand particles is observed i.e. the AA’ line in Figure (4.1). By finding where the AA’ line corresponds we converted the 560x160 matrix into a vector in the size of 560x1 each of the rows of the vector representing different inlet conditions. Then the density distribution is converted into the volumetric fraction by using Equation (2.9). Afterward, we obtain the number of sand particles per second by using the mean diameter of the sand particles, and the bulk velocity of the flow. In the below graph, both the volumetric fraction of particles and the number of particles injected per second are given for the case that the particles were injected in 560 different locations,

![Graph showing volumetric fraction distribution and number of particles per second](image)

**Figure 4.2:** Volumetric fraction distribution on the left, Number of particles per second on the right for 560 different inlet conditions.
Although, this process yielded us ejected number of particles per second it yields 560 different locations at which the injection rates of sand particles are different, which makes the modeling of the flow complicated. Therefore, by averaging the density matrix over 16x16 square matrices, we obtained a 35x10 matrix to model the multiphase flow. Likewise, in the process we did for the 560x160 matrix, the 35x10 matrix was also converted into a 35x1 vector. For the case where we have 35 different inlet conditions, the volumetric fraction ratio and the number of particles are given below,

\[ V_f \approx 10^{-6} \]

As discussed in

\textbf{Figure 2.3:} Volumetric fraction distribution on the left, Number of particles per second on the right for 35 different inlet conditions.

Although we compromised the resolution of the inlet condition, when we divided the inlet into 35 segments, this made modeling easier than the system with 560 inlets. The Figure (4.3) shows that the volumetric fraction reaches around the value of $10^{-6}$. As discussed in
Chapter (2.3), Figure (2.1) volumetric fraction of the order of $10^{-6}$, corresponds to the one-way coupled region. This means that the change in the flow of carrier fluid affects the flow of the dispersed phase, but the opposite is not true. Also in the one-way coupled case, the particle-particle collisions can be neglected. Further information can be found in the work of Elghobashi S., (2006) [6].

The decision of solving this multiphase flow can also be justified by looking at flow coupling parameters given by Michealides E. E., et al. (2023) as shown below. [5].

Since we do not have phase changing in the given flow conditions, the mass coupling ratio is equal to zero. For the momentum coupling from Equation (2.24), we can write,

$$\Pi_{mom} = \frac{3\pi n \mu L \alpha D}{\alpha C \mu_a u} \left(1 - \frac{v}{u}\right) \tag{4.4}$$

In which, n is the number of particles, L is the length of the channel where we are concerned with the multiphase flow, $\mu_C$, $\alpha_C$, $\rho_C$ are dynamic viscosity, volumetric fraction and material density of air, D is the mean diameter of particles v is conveying velocity of sand, u is the velocity of air.

As per the data we obtained and shown in Figure (4.3), we obtain the below value for the momentum coupling parameter,

$$\Pi_{mom} \approx 10^{-6} \tag{4.5}$$

As per reference [5],

$$\Pi_{en} \approx \Pi_{mom} \tag{4.6}$$

Therefore, we conclude that the multiphase flow we are interested in, can be regarded as one-way coupled. This situation makes the problem significantly simpler since we do not need to consider turbulence modulation due to particle-fluid interactions and the particle-particle collisions.

In addition, from Figure (4.3), we can infer that the average number of particles per second per the inlet is 2156. We will use this information when we are solving sand particle-laden airflow for the uniform loading case.
5. **Numerical Solution to the Multiphase Flow**

The numerical solution to the multiphase flow, in our specific case, sand particle-laden air flow over a compressor blade, has been obtained by OpenFOAM. OpenFOAM is as stated earlier an open-source software that is composed of modules written in C++ codes, which has the ability to solve continuity, Navier-Stokes, and energy equations of fluid dynamics by applying a finite volume method scheme. SolidWorks and Salome were used for the geometry creation and mesh generation. SolidWorks is a commercial solid modeling computer-aided design software, and Salome is an open-source software that provides the environment for the geometry creation and mesh generation.

5.1 **OpenFOAM Overview**

OpenFOAM stands for “Open Field Operation and Manipulation”. It is an open-source software produced by OpenCFD Ltd. company. The essential folders of OpenFOAM include the application folder, src folder, and tutorial folder.

The application folder includes the solver folder, utilities folder, tools folder, and test folder. The solver folder allows users to access and modify the solvers to a specific problem based on the users’ needs. The utilities folder includes tools pertaining to pre- and post-processing applications and mesh generation.

In the src folder, one can find several functions to modify the test case to obtain some desired quantities pertaining flow which may not be inherently solved by OpenFOAM solvers.

In the tutorial folder, there are pre-configured folders for solving incompressible, compressible flows, combustion, and multiphase flows, or folders for solving flow problems in the Lagrangian framework. In this thesis, we intend to solve a flow problem for immiscible gas-solid flow from the perspective of the Lagrangian framework. Two different solvers can solve this problem. One of them is reactingParcelFoam and the other one is simpleReactingParcelFoam.
Those solvers are defined in the reference [20] in detail. Below an overview of the main characteristics of these solvers is provided.

**“reactingParcelFoam**
Transient solver for compressible, turbulent flow with a reacting, multiphase particle cloud, and surface film modeling.

**simpleReactingParcelFoam**
Steady-state solver for compressible, turbulent flow with reacting, multiphase particle clouds and optional sources/constraints.”

In addition to these solvers’ ability to solve multiphase flow, both reactingParcelFoam and simpleReactingParcelFoam are able to handle chemical reactions and heat release due to combustion. To find the solution to sand particle-laden airflow, we will use simpleReactingParcelFoam to solve the multiphase problem for the steady-state case for simplicity. In the following chapters, we will elaborate on the solver folder and the case folder that we are going to use to solve sand particle-laden airflow.

### 5.2 The “simpleReactingParcelFoam” Solver

The simpleReactingParcelFoam solver can be found under the local OpenFOAM directory, in our case it is openfoam2306, since we will use version 2306 in the solution of this problem, openfoam2306\applications\solvers\lagrangian\reactingParcelFoam directory. This solver solves the multiphase problem by utilizing the SIMPLE (Semi Implicit Method for Pressure-Linked Equations) algorithm developed by Patankar & Spalding (1972) [21].

The SIMPLE algorithm is a pressure corrector method that basically estimates an initial pressure field and then solves x-momentum and y-momentum equations to find u and v velocities, for a 2D case. Then, the algorithm uses u and v velocities to obtain a pressure correction field with the help of the continuity equation. Given that the pressure correction field is determined, velocity correction fields are also identified by using pressure correction. In the last step pressure correction is used as the initial pressure field and the above steps are repeated
until the algorithm reaches a solution where the continuity equation is satisfied within a tolerance range. Further information can be found in reference [21].

As for solver directory, an OpenFOAM solver is generally comprised of the C++ file that solves the given flow problem whose name is identical to the solver, and the createFields header file to describe the scalar fields such as pressure, density temperature, etc., or the vector fields such as velocity that delineates the initial conditions, and the make folder to modify and compile the solver.

In our case, simpleReactingParcelFoam includes Make folder, createClouds, createFieldRefs, createFields, simpleReactingParcelFoam.C, EEqn, pEqn, UEqn, Yeqn files. The content of simpleReactingParcelFoam.C executable C++ file is given below,

```cpp
Info<< "\nStarting time loop\n" << endl;
while (simple.loop())
{
  Info<< "Time = " << runTime.timeName() << nl << endl;
  parcels.evolve();
  // --- Pressure-velocity SIMPLE corrector loop
  {
    #include "UEqn.H"
    #include "YEqn.H"
    #include "EEqn.H"
    #include "pEqn.H"
  }
  turbulence->correct();
  runTime.write();
  runTime.printExecutionTime(Info);
}
Info<< "End\n" << endl;
return 0;
```

**Figure 3.1:** The part of the simpleReactingParcelFoam.C file where the equations are solved.

We see that under the SIMPLE algorithm loop, the momentum equation is solved to determine U velocity under the UEqn.H file, and the energy equation is solved under EEqn. H file and YEqn file are utilized to determine phase changes and combustion that may occur in the
flow. The pEqn.H file is used for updating the U velocity field such that the continuity equation is satisfied as per the SIMPLE algorithm that is briefly described above.

Also, the simpleReactingParcelFoam.C file includes some include commands which help us to include the variables and classes defined in other files that are generally can be found under the src directory to consider turbulence, radiation models, and so on.

In the createFields.H file, one can find the fields that are required to describe the flow problem. For instance, Figure (5.2) which is cited from createFields.H under the simpleReactingParcelFoam solver shows us how to define density, velocity, and the composition of species.

Figure 5.2: The part of the createFields.H file that contains the information about variable addressing.

Lastly, the make folder of a solver consists of the “files” file and the “options” file. In the “files” file the name of the solver and its address are written. In the “options” file, all the files that are employed by the solver to solve the problem are supposed to be written with their location.
It is also possible to modify the solvers with the help of the Make folder. After the changes related to the solver are made by modifying createFields or the solver file, the solver files that contain additional files need to be addressed in the “options” file under the Make directory. After that by writing the wmake command to the terminal the new solver can be compiled.

5.3 The “simpleReactingParcelFoam” Tutorial

The pre-configured case that can be solved by the simpleReactingParcelFoam solver can be found in openfoam2306\tutorials\lagrangian\simpleReactingParcelFoam address. Initially developed case for this solver, solves compressible turbulent air-water mixture flow in a vertical channel for steady-state case. Our goal is to modify this solver to solve air-sand mixture flow.

The tutorial folders of OpenFOAM consist of an initial time folder, a constant folder, and a system folder. Even if we solve a steady-state flow, an OpenFOAM case has to be started, therefore in the initial time folder, i.e. “0” directory, the initial conditions for scalar and vector fields needs to be addressed.

In the “0” directory, the fields that need to be specified change depending on the flow problem that we are interested in. For the laminar incompressible flow, it is generally sufficient to specify only the pressure and velocity field whereas, for the compressible, turbulent, multiphase flow, the temperature that is required to solve the energy equation, turbulence model, and the mass fraction of species needs to be specified for the initial condition.

In the constant directory, one can find the files related to the properties of flow. The transport properties or the turbulence properties of flow can be found in the constant directory. For the compressible flow case, in the constant folder, the thermophysicalProperties file also exists to inform the solver which closure equations are going to be considered to determine the viscosity of flow, the specific heat capacity of flow, or other properties.

Since we are solving the flow from the Lagrangian framework, another file, reactingCloud1Properties, relating to track particles also exists in the constant folder. In the
reactingCloud1Properties file, one can find the entries to determine the properties of particles. In this file, it is possible to set the flow uncoupled, i.e. one-way coupled, or coupled. The inlet injection conditions such as the particle velocity in terms of parcel per second can also be determined. The effect of both particle-particle and particle-wall collisions can be counted by writing codes in this file.

Apart from that, in the constant file one can also find the polyMesh folder that contains the mesh information. The polyMesh folder stores mesh information under boundary, faces, neighbor, owner, and points files. The last folder that is in the case folder is the system folder. The main files in the system directory are controlDict, fvSchemes, and fvSolution.

In the controlDict file, one can specify the initial time at which the solver starts to solve and the end time. Also, the maximum CFL number can be introduced with the adjustable time step options to ensure that the problem is solved within stability limits.

In the fvSchemes file, the numerical schemes that are applied to the finite volume mesh elements to solve the governing equations of the flow, which is addressed in the solver folder, are specified.

In the fvSolution file, the matrix solution methods such as Gauss-Seidel methods are specified to solve the fields such as velocity, pressure, etc. It is possible to introduce relaxation factors to accelerate the convergence.

### 5.4 Introducing “sand” as a Solid Species to OpenFOAM

Before we start to investigate each folder that constitutes the tutorial simpleReactingParcelFoam, we need to explain how new species can be added to the tutorial case for solving the multiphase flow that is formed by species that are unknown to OpenFOAM.

The initial case that can be solved by simpleReactingParcelFoam is vertical channel flow that can be found in openfoam2306\tutorials\lagrangian\ simpleReactingParcelFoam
verticalChannel. However, this case includes air and water as species. Therefore, we need to create a new case folder.

To solve a multiphase flow for sand-air mixture, the species inside the “0” folder need to be recognized by the solver i.e. “simpleReactingParcelFoam”. The thermophysical properties of air are already known by the solver but we need to introduce the “sand” variable to OpenFOAM.

To do so, one needs to modify the source code that controls the thermophysical properties of species. One can access these source codes in openfoam2306\src\thermophysicalModels\thermophysicalProperties\solidProperties. In this folder, one can see that only carbon, ash, and calcium carbonate are specified as solid species. One can start to create the sand particle species by copying the ash folder and renaming it sand. The resulting sand folder contains ash.C and ash.H files. After renaming both files as sand.C and sand.H respectively, the below modifications need to be made to create the sand folder. For the sand. H file we need to plug in sand entries in ash entries as shown below Figure (5.3) on page 45,

```
class sand
{
    public : solidProperties

public:
    // Define type informations
    TypeName("sand");

    // Constructors
    // Construct null
    sand();
    // Construct from dictionary
    sand(const dictionary& dict);

    // Construct and return clone
    virtual autoPtr<solidProperties> clone() const
    {
        return autoPtr<solidProperties>::NewFrom<sand>(*this);
    }

    // I-O
    // Write the function coefficients
    void writeData(Ostream& os) const;

    // Ostream Operator
    friend Ostream& operator<<(Ostream& os, const sand& s);
};
```

*Figure 5.3:* The part of sand. H file that shows the entries that need to be changed.
For the sand C file, the modification needs to be applied as shown in Figure (5.4).

```cpp
// * * * * * * * * * * Member functions * * * * * * * * * * *
void Foam::sand::writeData(Ostream& os) const
{
    solidProperties::writeData(os);
}

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
```

**Figure 5.4:** The part of sand C file that shows the entries that need to be changed.

The values inside the solidProperties entry indicate the values of density (kg/m³), specific heat capacity (J/kgK), thermal conductivity (W/mK), the heat of formation (J/kg), emissivity, molar weight (kg/kmol), Poisson ratio, and the Young modulus of sand (N/m²), respectively. In this thesis, we did not take into account the effects of Young modulus and Poisson ratio. The thermal conductivity of sand particles was chosen congruent to the given data in the article written by Chen S. X., (2007) [22] and the specific heat of sand was chosen congruent to the given data in the article written by Baumann T., Zunft S., (2015) [23].

After these modifications are done, one needs to modify the Make folder which is accessible at openfoam2306\src\thermophysicalModels\thermophysicalProperties\Make. In the Make folder, in the “files” file, the below code that is shown in Figure (5.5) needs to be added.
Lastly, one can update the thermophysicalProperties library by writing the below codes into the terminal,

- cd openfoam2306/src/thermophysicalModels/thermophysicalProperties/
- wmake

After the library is compiled without errors, one can introduce sand particles into the tutorial case.

5.5 Geometry Creation and the Mesh Generation

To create a compressor blade profile, we utilized the information given in reference [24]. Aungier, R.H. (2003) stated that the existing convention of the United States is the use of NACA 65 profiles as compressor blade profiles. In contrast, British practice is often centered on using circular profiles as compressor blades [24].

In our consideration of selecting the compressor blade, we selected NACA 65-(10) 10 profiles, as this profile was once popularly used in the manufacturing of compressor blades. The first digit after NACA, which is 6, indicates the maximum camber as the percentage of the chord.
length. The second digit, i.e., 5, shows the position of maximum camber divided by ten concerning the chord length, so 5 indicates that maximum camber occurs at the 50% of the chord, the number in parenthesis indicates the lift coefficient in tenths, so (10) means that the lift coefficient is 1. The last two digits indicate the maximum thickness concerning chord length.

In reference [24], it is also noted that the lift coefficient is correlated with the maximum camber angle with the expression given below,

\[
\tan (\theta/4 ) = 0.1103xCl
\]  \hspace{1cm} (5.1)

Since accessing the compressor blade geometry information used in the M250 turboshaft engine was not viable, we could not obtain the maximum camber angle for the compressor blade geometry considered in this thesis. Therefore, to simplify the problem, we decided to use the NACA 6510 profile that only deems the maximum camber, maximum camber location, and blade thickness.

The compressor geometry based on NACA 6510 profile was created by using the tool that is accessible at Theory [http://airfoiltools.com/airfoil/naca4digit](http://airfoiltools.com/airfoil/naca4digit). A NACA 6510 profile was drawn with 200 points, and then the points of the blade profile were imported into an Excel file. This Excel file was then imported into SolidWorks to draw geometry using the option “Insert-\>Curve-\>Curve Through XYZ Points”. After inserting the NACA 6510 profile into SolidWorks, the chord length was set to 63 mm, as discussed in Chapter (4.3). Then, the geometry was installed as a stl file. The extension of stl stands for “STereoLithography,” also called “Standard Triangle Language,” a file format that describes the geometry with triangular meshes. In the STL files, the vertices of triangles and the normal vectors perpendicular to the surface are stored.

After obtaining the “stl” format of the geometry, two options exist to import geometry into OpenFOAM. One way to import the geometry into OpenFOAM requires using OpenFOAM meshing utilities. The other way requires using software for meshing the geometry, such as Gmsh, Salome, etc., and importing the meshed files into OpenFOAM. For the first option, the snappyHexMeshDict dictionary can convert the STL file into meshed geometry. The
snappyHexMeshDict works by removing the cells to obtain the desired mesh. For this reason, the entire computational domain must first be meshed, and then the geometry needs to be imported. Then, by defining the desired meshes to be removed, one can obtain an airfoil geometry for the compressor blade such that the surrounding of the blade is meshed.

To run snappyHexMesh, a user requires the following,

- An STL file that defines the geometry, located in the constant/triSurface subdirectory of the case directory,
- A background mesh that covers the whole computational domain located in the system directory of the case,
- The snappyHexMeshDict dictionary, with appropriate entries, is located in the system directory of the case,

For further information about snappyHexMeshDict, the OpenFOAM user guide can be utilized [25]. However, the usage of snappyHexMeshDict is somewhat complicated. Therefore, we created the geometry and mesh in another way. After creating geometry as an STL file, we imported the geometry into Salome, which is a software for geometry creation and mesh generation, to mesh the geometry.

Referring to Chapter (4.4), we know that the volumetric fraction of the dispersed particles varies depending on the location. Therefore, we decided to define four different inlet conditions, each with its flow properties such as volumetric fraction rate, number of injected particles per second, etc. For the 35 different inlet conditions discussed in section 4.4, we found that in the region that starts from y= 6 cm and ends at y=1.2 cm, the number of injected particles per second is relatively uniform, roughly 1131 particles per second. From y=1.2 cm to y=0 location, the sand particle loading is high with a mean value of 6163 particles per parcel per second. From y=0 cm to y=−4.8 location, the sand particle loading fluctuates, and the mean value of 2001 particles per parcel per second. The last region contains the particles from y=−4.8 to y=−6; particles in this region are uniformly distributed identically to the first region.
By considering the above inlet conditions variation, the below geometry is created in Salome,

![Image]

**Figure 5.6:** The drawing of the compressor blade profile that has 4 different inlet conditions, each color represents different inlet conditions.

After creating the geometry, the geometry is meshed using unstructured triangular meshes. For the mesh creation, the NETGEN 2D algorithm was utilized, for further information the readers referred to reference [26]; for 1D, the wire discretization scheme was applied along the x-axis. Then, the meshed geometry is converted into 3D geometry by extruding the 2D geometry along the z-axis, such that the blade has a span length of 6 cm. However, we intend to solve this problem for 2D geometry; therefore, the span length does not have much importance if we can identify the initial and boundary conditions appropriately.

With the help of Salome, we created 3 different meshes for the numerical investigation of the erosion of the compressor blade. Our intention of creating 3 mesh levels is to check whether the results we obtained converges as we refine the mesh.

For the finest mesh that we used to find the solution for multiphase flow around the compressor blade, the mesh ended up having a total number of 289417 faces, 132 of which are the faces that form the compressor blade profile.
After the mesh generation, the mesh file was imported into the OpenFOAM case as a “unv” file. Then, by writing the below codes to the terminal, one can convert the “unv” file to a mesh file format that OpenFOAM can process,

- ideasUnvToFoam name_of_the_file.unv

For checking the mesh quality, one can write checkMesh to the terminal to check the maximum skewness and the aspect ratio of the cells.

### 5.6 The “0” Directory of the Tutorial Folder

Finally, given that we briefly explained the geometry creation and introduction of the sand particles to the OpenFOAM environment, we can investigate the directories of the case tutorial more thoroughly. The “0” directory of the case that we created for air-sand mixture contains the following files as variables: air, alphat, k, nut, omega, p, sand, T, U. In which air file includes air mass fraction at different boundary conditions, alphat stores initial conditions for kinematic turbulent thermal conductivity, k represents turbulent kinetic energy, nut again stores the initial conditions for turbulent viscosity, omega stores values for specific dissipation rate, p stores the absolute pressure value of the boundary condition; likewise air sand stores the mass fraction of sand particles, T stores the temperature values and U file stores the bulk velocity of the flow.

For the uniform sand distribution case, from the graph given in Figure 4.3, we obtained that the volumetric fraction of sand particles for the uniform case is 2.19x10⁻⁷. If we use Equation 1.12, we obtain the mass fraction of sand particles as,

\[ Y_s = 4.84 \times 10^{-4} \]  

\[ Y_a = 0.9995 \]  

Hence the air and sand files were modified to meet the above criteria,

As an example, the air file is shared below,
Figure 5.7: The “air” file located in “0” folder, entries from “BackFace” to “Inlet4”
In the above file, BackFace and FrontFace represent the region in the xy plane in which we are not concerned with the solution of the flow in the line that is perpendicular to both BackFace and FrontFace since we are modeling our problem in 2D space. Therefore, the type of BackFace and FrontFace were set to empty. The “Upperwall” and “Lowerwall” patches represent the boundary of the numerical domain in the xz plane. 4 different inlet conditions, Inlet1, Inlet2, Inlet3, Inlet4, were set to model variable sand distribution. However, we intend to investigate both the uniform sand distribution case and the variable sand distribution case. For the uniform case, the air mass fraction was set to 0.9995 for every defined boundary condition.

5.7 The “constant” Directory of the Tutorial Folder

In the constant directory, one can find files related to the geometry definition, or files related to the turbulence model and thermophysical properties of multiphase flow components, and other files. In the “polyMesh” folder located in the constant folder, there exist files to store boundaries of the meshes, and the nodal points of the finite volume element.

For the compressible flow solvers, there is also “thermophysicalProperties” file where one can define the properties of air and sand particles. The user can use Sutherland Equations for viscosity or can opt to use either Perfect Gas Equations or polynomial approaches to estimate the specific heat constant of the materials by changing the entries of “thermophysicalProperties” file.

Apart from these files, One of the most important files in this directory is called “reactingCloud1Properties”. This file contains information about the particle injection model, physical properties of the dispersed particles, particle diameter distribution, the force model that both acts on particles, or the forces due to particle-particle collision and so on. The “reactingCloud1Properties” file can also calculate, some properties such as collision density of the particles when appropriate functions added to “reactingCloud1Properties” file. For the sake of brevity, we only would like to share some of the entries of “reactingCloud1Properties” file as below,
Figure 5.8: The “reactingCloud1Properties” entries related to force and injection model.

The green entries in Figure (5.8) shows the forces that we consider, which is drag force calculated based on drag coefficient proposed by Putnam A. (1961), and the gravitational force in which buoyancy is included. The orange entries show the number of particles and the parcel per second for the “Inlet1” condition, for uniform sand loading case. The yellow entries show the binned particle probability distribution, such that 10% of the particles are smaller than 50 µm and 45% of the particles are smaller than 210 µm. These values for particle distribution are obtained from the reference Ph.D thesis [1].
**Figure 4:** Some functions that were used for obtaining some useful data.

In Figure (5.9) “particleTracks1” entries allow us to track the parcels from the standpoint of the Lagrangian framework so that we could monitor the particle trajectories around the compressor blade. The “particleErosion” entry calculates the erosion by using Finnie Model, and it creates a “reactingCloud1Q” file under the time directories, so that the user could see which portion of the blade is more eroded compared to the other portions [15]. If it is not stated, the source code that controls “particleErosion” entry assumes that the K and $\psi$ are both equal to 2. To have more data, one could also add a “patchCollisionDensity1” entry to monitor the number of particle wall collisions per unit area.
5.8 The “system” Directory of the Tutorial Folder

In this folder, one could find a “controlDict” file which contains information about the simulation starting and ending time, or restrictions about maximum allowed CFL number and so on. The entry of this file is shown below,

```plaintext
application simpleReactingParcelFoam;
startFrom startTime;
startTime 0;
stopAt endTime;
endTime 300;
deltaT 1;
writeControl timeStep;
writeInterval 1;
purgeWrite 0;
writeFormat ascii;
writePrecision 10;
writeCompression off;
timeFormat general;
timePrecision 6;
runTimeModifiable yes;
```

*Figure 5.10:* Some entries of “controlDict” file located in “system” directory.
6. Results and Discussion

The erosion of the compressor blade, with a chord length of 63 mm, assumed to have a shape of NACA 6510 profile, was numerically investigated by using the OpenFOAM solver, which employs the finite volume control method.

A solution to sand particle-laden air flow for both the uniform sand loading case and the variable sand loading case for varying angles of attack for a compressible turbulent flow was found by considering steady-state conditions. In this context, the uniform sand loading case means that the number of particles in each cell are equal, so the sand particles are injected into the engine with a uniform mass loading. On the other hand, the non-uniform case represents the varying mass loading case, and the sand particle volumetric fraction distribution varies as per Figure (4.3).

In the consideration of finding a steady-state solution to the sand particle-laden airflow, it was considered that the airflow was continuously supplied by sand particles so that the sand concentration inside the mesh volume elements was constant. We should point out that the consideration of the steady-state case may not be the most accurate representation of sand particle-laden airflow since the sand particle concentration calculated inside the volume elements could be significantly changed due to the turbulent flow condition. However, we solved the problem for simplicity as if the flow is in a steady-state condition.

Only the gravity, buoyancy, and drag force were considered for the forces that act on the particles. Since we concluded that the multiphase flow can be regarded as dilute flow, we neglected the effects of discrete phase flow on the carrier flow and the particle-particle interactions. We also neglected the heat transfer via radiation.

For the erosion calculation, we utilized the “particleErosion” dictionary that can be found at openfoam2306\src\lagrangian\intermediate\submodels\CloudFunctionObjects. This model calculates the eroded volume based on the Finnie Model developed by Finnie I., (1960) [15].
Under these considerations, we obtained the eroded volume per unit span of the compressor blade both for the uniform sand loading case and variable sand loading case. The simulation was run to simulate 300 seconds of sand ingestion. However, since we solved the problem for the steady-state case, the simulation indeed solves the governing equations to access a steady-state condition in 300 steps rather than 300 seconds. A better approach could have been obtained by solving this problem for the transient case. However, while we were trying to solve this problem for the transient case, we encountered some coding issues, so we were not able to solve the multiphase flow problem for the transient scheme.

The sand particle-laden flow problem was solved for variable and uniform sand loading cases for the finest, medium, and coarsest mesh under different angle of attack conditions. The finest mesh contains 172076 2D face elements, 132 of which belongs to blade surface, including both triangles; the medium mesh contains 44892 2D face elements 50 of which belongs to blade surface, including both triangles, and the coarsest mesh has 10378 including 2D face elements, 19 of which belongs to blade surface.

6.1 Interpretation of Eroded Volume Results for Uniform Sand Distribution Case

While refining the mesh we tried to obtain a refinement factor of 2 in each direction i.e. the -x axis direction and the -y axis direction. Obtaining the results for the finest mesh for a single case took 2674 seconds, which corresponds to roughly 45 minutes. The problem was solved by using the medium mesh level within 1172 seconds. Solving the coarsest mesh, on the other hand, took 190 seconds.

The eroded volume per unit span was calculated using the Finnie Model (1960) for the uniform and non-uniform sand loading case for varying angle of attack for three different mesh levels [15]. For three mesh levels erosion due to uniform sand loading case are given below. For the figures in this section, “reactingCloud1Q” represents the eroded volume per unit span in (m³/m).
Figure 6.1: Eroded volume of compressor blade suction side for the finest mesh level in uniform case.

Figure 6.2: Eroded volume of compressor blade pressure side for the finest mesh level in uniform case.
**Figure 6.3:** Eroded volume of compressor blade suction side for the medium mesh level in uniform case.

**Figure 6.4:** Eroded volume of compressor blade pressure side for the medium mesh level in uniform case.
**Figure 6.5:** Eroded volume of compressor blade suction side for the coarsest mesh level in uniform case.

**Figure 6.6:** Eroded volume of compressor blade pressure side for the coarsest mesh level in uniform case.
The results we obtained and presented above show that the leading edge is more eroded compared to the trailing edge. Also, it is observed that the suction side of the compressor blade, which is the upper half of the blade, is much more eroded than the pressure side of the compressor blade, which is the lower half of the blade.

We assume that the leading edge is more susceptible to being eroded because the sand particles impacting the leading edge are scattered and may not be as much as likely to impact the trailing edge as they impact the leading edge. Likewise, the suction side is more eroded because of the higher velocities in the core potential flow outside the boundary layer. In future work the effect of different camber of the airfoil will be considered also, to gain perspective of how the local acceleration of the potential flow affects erosion rates.

When we look at the medium mesh level case erosion that happened over the suction side, we can also detect that the suction of the side of the trailing edge is also vulnerable to erosion. Although it is not predicted that the trailing edge will be eroded as much as the leading edge, erosion on the trailing edge may cause severe problems in aircraft operation since the trailing edge is thinner, which may lead to blade ruptures. Tabakoff W., Hamed A., and Metwally M. (1991) also stated the importance of trailing edge erosion [27].

The results that show the leading edge is more likely to be eroded compared to the trailing edge are also congruent with the findings of Grant G., Tabakoff W., (1975) [28], Tabakoff W., Hamed A., Metwally M., (1991) [27].

6.2 Interpretation of Eroded Volume Results for Non-uniform Sand Distribution Case

In this master’s thesis study, the effects of the variable volumetric fraction case of sand particles on compressor blade erosion were also investigated for the finest, medium, and coarsest mesh cases. The results for this case are given below,
Figure 6.7: Eroded volume of compressor blade suction side for the finest mesh level in non-uniform case.

Figure 6.8: Eroded volume of compressor blade pressure side for the finest mesh level in non-uniform case.
Figure 6.9: Eroded volume of compressor blade suction side for the medium mesh level in non-uniform case.

Figure 6.10: Eroded volume of compressor blade pressure side for the medium mesh level in non-uniform case.
**Figure 6.11:** Eroded volume of compressor blade suction side for the coarsest mesh level in non-uniform case.

**Figure 6.12:** Eroded volume of compressor blade pressure side for the coarsest mesh level in non-uniform case.
The results obtained for the non-uniform sand loading case are similar to the uniform sand loading case; both of them verify that the leading edge is more susceptible to erosion than the trailing edge.

An interesting feature of the numerical study shows that for both the uniform and non-uniform cases, the finest and the coarsest mesh can capture the erosion on the pressure side of the blade; however, medium mesh level cannot detect erosion over the pressure side of the compressor blade. The reason for that is that the finest mesh, as expected, solves the problem more rigorously in terms of reducing discretization errors so that it can capture the eroded volume. On the other hand, for the coarsest mesh case, the blade profile was divided into fewer parts so it could capture the eroded volume over the pressure side as an extension of the suction side part of the leading edge.

6.3 The Comparison of Uniform and Non-uniform Sand Loading Case

For the sake of brevity, we will only compare the suction side of the compressor blades for the finest mesh. The results for both cases are given below.

![Figure 6.13: Eroded volume of compressor blade suction side for the finest mesh level. Non-uniform sand mass loading case is on the left, Uniform sand loading case is on the right.](image)
Figure (6.13) shows that when the sand mass flow is non-uniform in terms of volumetric fraction, the compressor blade is eroded more continuously. Furthermore, the maximum affected region due to erosion shifts toward the trailing edge in the case of non-uniform sand loading. Also, the severity of erosion increases with the non-uniformity of sand loading distribution. We found that the maximum eroded volume in the case of non-uniform loading is $1.26 \times 10^{-10} \, \text{m}^3$ per unit span, whereas the maximum eroded volume for the uniform case is $8.11 \times 10^{-11} \, \text{m}^3$ per unit span.

We estimate that the reason for this, in the non-uniform case, as seen in Figure (4.2) and Figure (4.3), is that the sand particles are concentrated at the center; therefore, they collide into the leading edge much more frequently, which may increase the eroded volume. Also, the fact that sand particles have higher concentration even in the 1.2 cm above the middle point of the inlet may have caused the eroded region to shift towards the trailing edge.

This situation can also be observed by looking at the collision density of sand particles for the uniform and non-uniform sand loading case. The number of collisions on the compressor blade per area is given in the Figure (6.14) below for the finest mesh.

**Figure 6.14:** Number of the particle collisions on compressor blade suction side for the finest mesh level, Non-uniform sand mass loading case is on the left, Uniform sand loading case is on the right.
6.4 The Evaluation of Eroded Volume for Different Angles of Attack In the Case of Stationary Compressor Blade

In this section, the eroded volume of the compressor blade was evaluated in terms of varying flow angles. We know that the multiphase flow travels through the x-axis from the leading edge of the compressor blade to the trailing edge of the blade. Also, the blade is rotating around the x-axis. Therefore, the multiphase flow interacting with the blades feels an angle of incidence or a flow angle due to the translational velocity of the compressor blades. The translational velocity of the compressor blade will increase with increasing span length of the blade, and this velocity can be expressed as below,

\[ V_b(l) = \frac{2\pi n l}{60} \]  \hspace{1cm} (6.1)

In the equation above, \( V_b(l) \) is the translational velocity of the compressor blades as a function of the distance between the collision location and the hub location, \( n \) is the number of revolutions of the compressor shaft per minute, and \( l \) is the distance between where the particles collided and where the compressor hub is located. If we consider that the air is inhaled into the engine with an angle of attack of \( \alpha \), the resultant angle of attack that the sand particles collide with the compressor blade will be the sum of this angle of attack, and the additional flow angle stems from the flow relative velocity to the compressor blade velocity. Since the flow angle would be a function of the distance between the location on the compressor blade at which sand particles have collided and the compressor hub, the incidence angle and the resultant angle of attack will increase with increasing distance. This resultant angle of attack can be expressed as below,

\[ \alpha_r = \alpha + \arctan \left( \frac{V_b(l)}{U} \right) \]  \hspace{1cm} (6.2)

Since we do not have information on the compressor geometry and the number of revolutions of the blades, the evaluation of compressor blade erosion with varying angles of attack was performed for the stationary case of the compressor blades, where \( n \) both zero. For this case, uniform and non-uniform case erosion was evaluated for 3°, 6°, 9°, 12°, 15°, and 18° degrees. The results for the uniform sand loading case are given below,
Figure 6.15: Eroded volume of compressor blade suction side for the coarsest mesh level in uniform case, $3^0$ degrees of angle of attack is on the left, $6^0$ degrees of angle of attack is on the right.

Figure 6.16: Eroded volume of compressor blade suction side for the coarsest mesh level in uniform case, $9^0$ degrees of angle of attack is on the left, $12^0$ degrees of angle of attack is on the right.
Figure 6.17: Eroded volume of compressor blade suction side for the coarsest mesh level in uniform case, 15° degrees of angle of attack is on the left, 18° degrees of angle of attack is on the right.

When we look at the eroded volume regions, we observe that the trailing edge is severely affected by erosion for both 3 degrees of angle and 18 degrees of attack. The findings show that the sand particles that rebounded from the engine case collided in similar locations for 3 degrees of angle and 18 degrees of attack. For the non-uniform case below, erosion results were obtained for the angle of attacks from 3 degrees to 18 degrees,

Figure 6.18: Eroded volume of compressor blade suction side for the coarsest mesh level in non-uniform case, 3° degrees of angle of attack is on the left, 6° degrees of angle of attack is on the right.
Figure 6.19: Eroded volume of compressor blade suction side for the coarsest mesh level in non-uniform case, 90 degrees of angle of attack is on the left, 120 degrees of angle of attack is on the right.

Figure 6.20: Eroded volume of compressor blade suction side for the coarsest mesh level in non-uniform case, 150 degrees of angle of attack is on the left, 180 degrees of angle of attack is on the right.
For the non-uniform case, it is also found that $3^0$ and $18^0$ degrees of angle of attack affect the similar regions in the trailing edge, which is noted that trailing edge erosion could cause severe problems since it is thinner compared to the leading edge by Tabakoff W., Hamed A., Metwally M. (1991) [27].

6.5 Mesh Refinement Analysis

In this section, we would like to present a qualitative mesh refinement study based on the 3 mesh levels that are introduced at the beginning of Chapter 6. We compared the total eroded volume obtained from 3 mesh levels for the uniform sand loading case in which the angle of attack is 0 degree, that is presented as column graph as below,

![Eroded Volume For Constant Sand Loading](image)

**Figure 6.21: Total eroded volume per unit span for 3 different mesh levels.**

As we refine the mesh, the total eroded volume per unit span decreases. We assume that this is because of the fact that as we refine meshes, the mass of particles that are colliding a specific region of compressor blade decreases and this causes decrease in the total eroded volume as per Finnie Model developed in 1960 [15].
7. Summary

We could summarize the total eroded volume data that we obtained for varying angle of attack, and the uniformity of the sand loading case with below graph,

![Graph showing total eroded volume under different conditions](image)

**Figure 7.1:** Total eroded volume under different conditions.

In the Figure (7.1), we see that the total volume of erosion is affected by both the angle of attack and the uniformity of the sand loading. As the angle of attack increases the eroded volume tends to increase since the particle velocities increase. However, at some angles such as angle of attack of 9 degrees, the total erosion is seen to be less than the total erosion at 0 degrees for non-uniform sand loading case. We assume that the particles tend to miss colliding the compressor blade at some angles when compared to the other angles. We could visualize this situation with below Figure (7.2),

72
Figure 7.2: Number of collisions per unit area for non-uniform sand loading case and the finest mesh, 0 degrees of angle of attack case is on the left, 9 degrees of angle of attack case is on the right.

In the Figure (7.2), for the 0 degrees of angle of attack particles are more likely to hit compressor blade which yields the maximum number of collision per unit area value of 44000 (1/m²) whereas, in the case of 9 degrees of angle of the maximum number of collision per unit area can reach around 42000 (1/m²).

Another interesting feature can be observed that as the angle of attack changes, the particles tend to impact similar regions periodically. This situation can be visualized as below,
Figure 7.3: Number of collisions per unit area for non-uniform sand loading case and the finest mesh, 0 degrees of angle of attack case is on the left, 9 degrees of angle of attack case is on the right.

In Figure (7.3), it is seen that for both 3 degrees and 18 degrees of angle of attack case, particles tend to impact near trailing edge, which may mean that as the angle of attack changes particle may tend to impact similar regions periodically because of that particles are rebounding from the wall and can be directed to the similar regions of the blade.
8. Conclusion

The effects of sand particle-laden airflow over the compressor blade on erosion due to sand ingestion have been investigated numerically for both the uniform and non-uniform sand loading cases for the varying angle of attack. A new flow model that allows the analysis of multi-dispersed clouds has been developed. A multi-injection region approach to simulate non-uniformity on the inflow sand content is investigated.

In the numerical solution of this multiphase flow, the flow is considered a steady-state, compressible, turbulent flow in which phase changes and chemical reactions do not occur. To solve this problem, we utilized OpenFOAM for the numerical solution of the problem, the features of which are summarized in Chapter 5.

In the solution, one-way coupled multiphase flow was solved for 2D compressor geometry whose chord length is 63 mm and has a shape of NACA 6510 profile. The solution was obtained by considering a single isolated compressor blade, therefore, the influence of adjacent blades were neglected for simplicity. For the turbulence model, the k-\(\omega\) model was utilized. Then, the multiphase flow was solved for three mesh levels, with a refinement factor of 2, for both uniform and non-uniform sand loading cases.

The initial conditions of sand volumetric fraction were taken from the Ph.D. thesis written by Olshefski K. T. (2023) [1]. The effects of these novel measurements on blade erosion have not been previously analyzed leading to a new insight on the effects of non-uniformity of erosion effects.

Although the numerical study as part of this master’s thesis considered steady-state conditions, one could obtain better results by solving this problem as a transient case. However, I encountered some convergence-related issues while solving the problem using the transient scheme.

In this thesis, it is shown that the leading edge is more susceptible to erosion than the trailing edge. Also, it is found that the non-uniform sand loading case increases the eroded
volume and makes the most eroded region shift toward the trailing edge. The result that shows the leading edge is more susceptible to erosion than the trailing edge is consistent with previous work reported in references [27,28]. The suction side of the airfoil is more susceptible to erosion because of the acceleration of the air in the potential core.

In addition, the effect of the angle of attack for the stationary case in which compressor blades are not rotating was investigated. There could be a trigonometric relation between the affected regions and the angle of attack. It is found that according to the angle of attack uniform or non-uniform sand loading may cause more erosion, as per Figure (7.1), for the specific volumetric fraction distribution considered in this master’s thesis.

In the future, the inlet conditions of the engine can be modeled with higher resolution. In this thesis, we considered four different inlet conditions while modeling the non-uniform sand loading case, which may have caused the discontinuity in calculating eroded volume.

Moreover, having more information on compressor blade geometry could lead to a more realistic representation of the erosion patterns. To foresee how the adjacent blades in a row affect the erosion of their neighbor blades, we need to have a cascade solidity ratio that relates the distance between the blades to the chord length of a blade.

Finally, this problem will be extended to the erosion modeling compressor blade in 3D, considering the variation of angle of attack due to the rotor speed. In this case, the angle of attack has to be regarded as a function distance between the hub and the location at which the sand particle collided.
References


