

Hydrodynamics of sorbents in a semi-dry circulating fluidised bed riser for flue gas purification

E Viljoen

 **orcid.org/0000-0002-1106-0047**

Dissertation accepted in fulfilment of the requirements for the degree *Master of Engineering in Chemical Engineering* at the North-West University

Supervisor: Mr AF van der Merwe

Co-supervisor: Prof RC Everson

Co-supervisor: Dr FH Conradie

Graduation: April 2024

ABSTRACT

Nowadays, the increasing stringent environmental regulations results in coal-fired power plants struggling to comply with these regulations. The attention toward technologies for the abatement of pollutant gases has increased significantly over the last few decades. The focus of the industry primarily lies with the removal of sulphur and carbon from the flue gasses being emitted to the atmosphere. Both Flue gas desulphurisation (FGD) and Carbon capture and storage (CCS) technologies are very mature technologies, but factors relating to gas-particle interactions are continuously researched in an attempt to further improve the efficiency thereof. Accordingly, research was done on a laboratory scale semi-dry circulating fluidised bed (CFB) using hydrated lime and potassium carbonate as sorbents respectively.

Although extensive research on both FGD and CCS technologies has been done in the past, limited information could be found pertaining to the direct influence of gas temperature and gas relative humidity as two independent variables on the hydrodynamics in a CFB. The aim of this study was to describe the hydrodynamics of hydrated lime and potassium carbonate under varying inlet gas temperatures as well as varying inlet gas relative humidity. The aim was achieved by describing the influence of both independent variables on two-phase flow in a CFB using both experimental results and numerical method CFB modelling.

The method chosen for increasing the inlet gas temperature and inlet gas relative humidity was adapted from literature to be suitable for this study. Operating parameters were chosen to ensure operation in the fast fluidisation regime to enable the ability to compare with past and future CFB research. Measured and calculated data responses were recorded experimentally and numerically, including pressure differentials, temperature profiles and local and average solids volume fraction at different riser sampling ports. Each response was addressed by discussing the influence of varying operating conditions and therefore an extensive amount of experimental work as well as CFD simulations were required.

The inlet gas temperature and inlet gas relative humidity were the varying parameters in this study. The inlet gas relative humidity showed the most significant influence between the two. Although captured slightly, the limited operating range of the equipment resulted in a less prominent influence than expected. The experimental results indicated a pressure differential across the riser with negative pressure readings as expected. It was also observed that preferential particle flow occurred near to the riser wall opposite the gas inlet. The inlet-outlet configuration of the riser could be the reason for this asymmetric flow profile.

In addition to the extensive amount of experimental work done in this study, CFD simulations were done to aid in discussions pertaining to the hydrodynamics in a CFB riser. The geometry of the riser was constructed using Siemens NX-12[®] software and simulations were carried out in STAR-CCM+[®] CFD software. Moreover, a Eulerian-Lagrangian with two-way coupling approach was followed. The average particle sizes were based on the D[4,3] retrieved from PSD analysis and the Schiller-Naumann homogenous drag model was employed.

The models were validated against experimental results and was found to successfully describe the hydrodynamics of the two sorbents in a CFB riser, despite the inability to capture the exact values. In addition, the aim was to investigate the influence of inlet gas temperature and inlet gas relative humidity on the hydrodynamics, and this was achieved using the experimental and modelling results.

Keywords: *CFD modelling, CFB, CCS, FGD, hydrodynamics, Hydrated lime, Potassium carbonate*

ACKNOWLEDGEMENTS

I would like to acknowledge my supervisor, Mr. Frikkie van der Merwe, for proving me with an endless amount of guidance, support and valuable inputs during the execution and writing of this dissertation. Furthermore, I would like to express my appreciation to my co-supervisors, Dr. Frikkie Conradie and Prof. Ray Everson for their guidance and time dedicated to this study.

Thank you to my peers, and the engineering workshop staff, for always assisting where needed and your ability to help making plans. Your efforts are deeply appreciated.

To my parents, and brother, thank you for your endless support and willingness to help when necessary. No word will ever describe my gratitude towards you during the past few years. Thank you for giving me the opportunity to pursue a degree in Chemical Engineering, without you this dissertation would not have been possible.

To my husband, Ruben Coetzer, thank you for your incredible support, help and motivation along the way and especially during this past year with writing this dissertation. I am thankful to have you by my side, encouraging me every step of the way.

Lastly, I would like to thank my Heavenly Father for blessing me with the opportunity to pursue this degree. Thank you for giving me the strength, guidance and ability to fulfil this chapter in my life. I will be forever grateful.

TABLE OF CONTENTS

ABSTRACT	I
ACKNOWLEDGEMENTS	III
LIST OF TABLES	IX
LIST OF FIGURES.....	XI
LIST OF ABBREVIATIONS	XV
NOMENCLATURE.....	XVII
CHAPTER 1 INTRODUCTION.....	1
1.1 Background and motivation	1
1.2 Research problem, purpose, and objectives	4
1.2.1 Research problem	4
1.2.2 Purpose of research	4
1.2.3 Objectives.....	5
1.3 Scope of study	5
1.3.1 Delineation of the study	5
1.3.2 Structure and layout of the dissertation.....	5
CHAPTER 2 LITERATURE REVIEW.....	7
2.1 Emission and emission control strategies	7
2.1.1 The current state of emissions in South Africa (SA)	7

2.1.2	Flue gas desulphurisation (FGD)	8
2.1.3	Carbon capture and storage (CCS)	11
2.1.4	The importance and review of research towards emission control strategies	13
2.2	Circulating fluidised beds (CFB)	14
2.2.1	Characteristics and operating conditions of CFBs.....	14
2.2.2	Hydrodynamics in circulating fluidised beds (CFB's).....	17
2.2.3	Application of CFBs in emission control technology	20
2.2.4	Evaluation of research towards CFB hydrodynamics	21
2.3	Computational fluid dynamic (CFD) modelling of circulation fluidised beds (CFB's)	24
2.3.1	Introduction.....	24
2.3.2	Numerical methods for modelling CFB hydrodynamics.....	24
2.3.3	Review of recent Eulerian-Lagrangian (E-L) and Eulerian-Eulerian (E-E) approach on CFB hydrodynamic studies	26
2.3.4	Closure laws.....	30
2.3.4.1	Drag closure model	30
2.3.4.2	Turbulence modelling of CFB's.....	31
2.4	Conclusion.....	32
CHAPTER 3	EXPERIMENTAL METHOD	34
3.1	Materials and sample properties determination	34
3.2	Experimental set-up	34
3.2.1	P&ID and riser dimensions	34
3.3	Experimental plan.....	38

3.4	Data collection	39
3.4.1	Calibration of gas inlet velocity	39
3.4.2	Calibration of sorbent mass flow rate	39
3.4.3	Temperature and relative humidity calculations	39
3.4.4	Experimental data.....	40
CHAPTER 4	CFD MODEL DEVELOPMENT	43
4.1	Model development	43
4.1.1	CFD model selection and considerations.....	43
4.1.1.1	Governing equations.....	43
4.1.1.2	Turbulence model.....	44
4.1.1.3	Drag model.....	46
4.1.1.4	Heat transfer model	47
4.1.2	Geometry discretisation and solution algorithm.....	48
4.2	CFD simulation procedure	51
4.2.1	Computer-aided design (CAD) of the CFB riser	51
4.2.2	STAR-CCM+® simulation procedure	52
4.3	Mesh independence analysis	54
CHAPTER 5	RESULTS AND DISCUSSION.....	59
5.1	Solid sorbent properties	59
5.2	Validation of CFD model with experimental data	60
5.2.1	Hydrated lime model validation	60
5.2.2	Potassium Carbonate model validation.....	64

5.3	Influence of operating parameters	69
5.3.1	Hydrated Lime	69
5.3.1.1	Influence of inlet gas temperature.....	69
5.3.1.2	Influence of inlet gas relative humidity	80
5.3.2	Potassium carbonate	88
5.3.2.1	Influence of inlet gas temperature.....	88
5.3.2.2	Influence of inlet gas relative humidity	96
5.4	Comparison of results for hydrated lime and potassium carbonate	103
CHAPTER 6	CONCLUSION AND RECOMMENDATIONS.....	107
6.1	Conclusion.....	107
6.2	Recommendations.....	110
BIBLIOGRAPHY		112
ANNEXURE A	CALIBRATION	124
A.1	Calibration data of the fan (FAN-01)	124
A.2	Calibration data of the sorbent feeder (FDR-01).....	126
A.3	Inlet gas temperature and relative humidity calculations.....	128
A.4	Inlet gas temperature and relative humidity measurements	131
ANNEXURE B	EQUIPMENT AND INSTRUMENTATION.....	133
B.1	General Information.....	133
B.2	Sorbent sampling procedure	134
ANNEXURE C	CONFIDENCE INTERVAL CALCULATIONS.....	136
C.1	Hydrated Lime (Ca(OH)₂) confidence interval.....	136

C.2	Potassium carbonate (K_2CO_3) confidence interval.....	140
ANNEXURE D HYDRATED LIME ADDITIONAL RESULTS		144
D.1	Experimental radial solids distribution for hydrated lime	144
D.2	Modelling radial distribution for hydrated lime	146
ANNEXURE E POTASSIUM CARBONATE ADDITIONAL RESULTS		148
E.1	Experimental radial solids distribution for potassium carbonate.....	148
E.2	Modelling radial solids distribution for potassium carbonate.....	150

LIST OF TABLES

Table 2-1: Commercial operating parameter ranges for CFB's	16
Table 2-2: Reported commercial CFB application worldwide	20
Table 2-3: CFB hydrodynamic studies where temperature and/or humidity is investigated	22
Table 2-4: Recent E-L and E-E CFB hydrodynamic studies	27
Table 3-1: Scientific standards used for particle size distribution (PSD) and density analysis	34
Table 3-2: Experimental variable conditions	38
Table 4-1: Initial gas and particle properties and constants used in STAR-CCM+® simulations	52
Table 4-2: Mesh properties for mesh independence analysis	54
Table 5-1: Solid sorbent properties	59
Table 5-2: Ca(OH)_2 – Percentage deviation of the modelled temperature from the experimental temperature and the R^2 -value for different operating conditions	62
Table 5-3: K_2CO_3 - Percentage deviation of the modelled temperature from the experimental temperature and the R^2 -value for different operating conditions	66
Table 5-4: Ca(OH)_2 - Experimental pressure differential across (i) the riser and (ii) the riser upper region under varying inlet gas conditions	69
Table 5-5: Ca(OH)_2 - Modelled static gauge pressure values for positions (a), (b) and (c) and modelled pressure differential across the riser	72
Table 5-6: Ca(OH)_2 - Experimental temperatures at different points across the riser for varying inlet gas temperature	73
Table 5-7: Ca(OH)_2 - Modelled temperatures at different points across the riser for varying inlet gas temperature	74

Table 5-8: Ca(OH)_2 - Experimental pressure differential across riser and riser upper region under varying inlet gas conditions	81
Table 5-9: Ca(OH)_2 - Modelled static gauge pressure values for positions (a), (b) and (c) on Figure 5.9 and modelled pressure differential across the riser.....	81
Table 5-10: Ca(OH)_2 - Experimental temperatures at different points across the riser for varying inlet gas conditions	82
Table 5-11: Ca(OH)_2 - Modelled temperatures at different points across the riser for varying inlet gas conditions	83
Table 5-12: K_2CO_3 - Experimental pressure differential across riser and riser upper region under varying inlet gas conditions	88
Table 5-13: K_2CO_3 - Modelled pressure at different points in the riser under varying inlet gas conditions	89
Table 5-14: K_2CO_3 - Experimental temperatures recorded at various points across the riser for varying inlet gas conditions	90
Table 5-15: K_2CO_3 - Modelled temperatures recorded at various points across the riser for varying inlet gas conditions	91
Table 5-16: K_2CO_3 - Experimental pressure differential across riser and riser upper region for varying inlet gas conditions	96
Table 5-17: K_2CO_3 - Modelled static gauge pressure and modelled pressure differential across the riser	97
Table 5-18: K_2CO_3 - Experimental temperatures at different points across the riser for varying inlet gas conditions	98
Table 5-19: K_2CO_3 - Modelled temperatures at different points across the riser for varying inlet gas conditions.....	99

LIST OF FIGURES

Figure 1-1: Layout of the dissertation	6
Figure 2-1: Schematic representation of a typical CFB configuration.....	16
Figure 2-2 Schematic of core-annulus flow pattern in CFB's.....	19
Figure 2-3: CFD model classification	25
Figure 3-1: P&ID of the experimental facility.....	35
Figure 3-2: Illustration to indicate main riser dimensions.	37
Figure 3-3: Schematic illustration of the different inlets, outlet, and sampling port arrangement.....	38
Figure 3-4: Location of radial sampling points	41
Figure 3-5: Schematic illustration of the non-isokinetic sampling probe	41
Figure 3-6: Measurement points for temperature, pressure and solids volume fraction across the riser	42
Figure 4-1: Different cell structures used in CFD discretisation: (i) Tetrahedral, (ii) Hexahedral and (iii) Polyhedral mesh structures	48
Figure 4-2: Flow diagram illustrating the SIMPLE algorithm	49
Figure 4-3: Adjusted SIMPLE algorithm.....	50
Figure 4-4: CFB Riser geometry.....	51
Figure 4-5: Percentage invalid cells present in each mesh	55
Figure 4-6: Percentage cells with good cell metrics in each category	56
Figure 4-7: Time required to simulate 5 seconds of riser operation for each different mesh	56
Figure 4-8: Modelled pressure at the top of the riser for each different mesh	57

Figure 4-9: Modelled average solids volume fraction at three riser ports for each different mesh	58
Figure 4-10: Mesh structure in CFD modelling	58
Figure 5-1: Ca(OH)_2 - Experimental and modelled pressure differential across the riser at different operating conditions	60
Figure 5-2: Ca(OH)_2 - Experimental and Modelled Temperature at four different point in the riser	61
Figure 5-3: Ca(OH)_2 – Modelled vs experimental radial solids volume fraction at the bottom port of the riser	63
Figure 5-4: Ca(OH)_2 – Modelled vs experimental radial solids volume fraction at the top port of the riser	63
Figure 5-5: K_2CO_3 - Experimental and modelled pressure differential across the riser at different operating conditions	65
Figure 5-6: K_2CO_3 - Experimental and modelled Temperature at four different point in the riser	66
Figure 5-7: K_2CO_3 - Modelled vs experimental radial solids volume fraction at the bottom port of the riser	68
Figure 5-8: K_2CO_3 - Modelled vs experimental radial solids volume fraction at the bottom port of the riser	68
Figure 5-9: Ca(OH)_2 - General static pressure profile across the riser	71
Figure 5-10: Ca(OH)_2 - General temperature profile of the riser for hydrated lime indicating the different temperature points.....	74
Figure 5-11: Ca(OH)_2 - Radial solids distribution profile at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas temperature	76
Figure 5-12: Ca(OH)_2 – Modelled radial solids volume fractions at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas temperature	78

Figure 5-13: Ca(OH)_2 - Experimental average axial solids distribution for varying inlet gas temperature.....	79
Figure 5-14: Ca(OH)_2 - Modelled average axial solids distribution for varying inlet gas temperature.....	80
Figure 5-15: Ca(OH)_2 – Experimental radial solids distribution profile at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet conditions.....	85
Figure 5-16: Ca(OH)_2 - Modelled radial solids volume fractions at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas conditions	86
Figure 5-17: Ca(OH)_2 - Experimental average axial solids distribution under varying inlet gas conditions	87
Figure 5-18: Ca(OH)_2 - Modelled average axial solids distribution under varying inlet gas conditions.....	87
Figure 5-19: K_2CO_3 – Experimental radial solids distribution profile at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet conditions.....	93
Figure 5-20: K_2CO_3 – Modelled radial solids volume fractions at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas conditions	94
Figure 5-21: K_2CO_3 - Experimental average axial solids distribution for varying inlet gas conditions.....	95
Figure 5-22: K_2CO_3 - Modelled average axial solids distribution for varying inlet gas conditions.....	96
Figure 5-23: K_2CO_3 - Radial solids distribution profile at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas conditions.....	100
Figure 5-24: K_2CO_3 – Modelled radial solids volume fractions at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas conditions	101
Figure 5-25: K_2CO_3 - Experimental average axial solids distribution for varying inlet gas conditions.....	102

Figure 5-26: K_2CO_3 - Modelled average axial solids distribution for varying inlet gas conditions.....	102
Figure 5-27: Pressure differential comparison of hydrated lime (HL) and potassium carbonate (PC) across the riser and riser upper region	103
Figure 5-28: Temperature comparison of hydrated lime (HL) and potassium carbonate (PC)	104
Figure 5-29: Radial solids distribution comparison of hydrated lime (HL) and potassium carbonate (PC) at the bottom port of the riser	105
Figure 5-30: Radial solids distribution comparison of hydrated lime (HL) and potassium carbonate (PC) at the middle port of the riser	105
Figure 5-31: Radial solids distribution comparison of hydrated lime (HL) and potassium carbonate (PC) at the top port of the riser	106

LIST OF ABBREVIATIONS

Abbreviation	Description
CCS	Carbon Capture and Storage
CBT	Carbon Tax
CFB	Circulating fluidised bed
CFD	Computational fluid dynamics
CPFD	Computational Particle Fluid Dynamics
CAD	Computer Aided design
DDPM	Dense Discrete phase method
DEM	Dense element method
DPM	Dense phase method
DNS	Direct numerical simulation
EMMS	Energy minimisation multi-scale
EOS	Equation of state
E-E	Eulerian-Eulerian
E-L	Eulerian-Lagrangian
FEM	Finite element method
FVM	Finite volume method
FGD	Flue gas desulphurisation
G	Global
GHG	Greenhouse gases
HL	Hydrated lime
KTGF	Kinetic theory for granular flow

Abbreviation	Description
LES	Large eddy simulation
L	Local
MP-PIC	MultiPhase-Particle-in-cell
MSW	Municipal solid waste
PDE	Partial differential equation
PSD	Particle size distribution
p-p	Particle-particle
PC	Potassium Carbonate
P&ID	Process and instrumentation diagram
RKE	Realizable k- ϵ
RH	Relative humidity
RANS	Reynolds-average Navier-stokes
SIMPLE	Semi-implicit method for pressure linked equations
SA	South Africa
SKE	Standard k- ϵ
D[3, 2]	Surface volume mean diameter (Sauter mean)
O-S	Syamlal O'brien
D[4, 3]	Volume mean diameter (De Brouckere mene)

NOMENCLATURE

GREEK LETTERS

Symbol	Description	Unit
$\bar{\varepsilon}_s$	Average solids volume fraction	-
μ	Dynamic viscosity	N·s/m ²
ε_d	Energy dissipation rate	m ² /s ²
ρ_g	Gas density	kg·m/s ²
α_g	Gas phase volume fraction	-
$\bar{\tau}_g$	Gas stress tensor	kg/ms ²
$\vec{u}_g$	Gas velocity	m/s
β_{fp}	Interphase momentum exchange coefficient	Ns/m ⁴
ν	Kinematic viscosity	m ² /s
ε_s	Local solids volume fraction	-
ρ_p	Particle density	kg·m/s ²
α_p	Particle volume fraction	-
τ_{ij}	Reynolds stress	m ² /s ²
μ_t	Turbulent eddy viscosity	m ² /s
k	Turbulent kinetic energy	J

ROMAN LETTERS

Symbol	Description	Unit
A	Area	m ²
$\vec{F}_B$	Body forces per unit volume	kg/m ² s ²

Symbol	Description	Unit
$C_1, C_2, C_\mu, \sigma_k, \sigma_\varepsilon$	Constants	-
$\vec{F}_{Con}$	Contact forces	kg·m/s ²
C_O	Courant Number	-
L	Distance between contact point and particle centroid	m
C_D	Drag coefficient	-
$\vec{F}_{Drag}$	Drag force	kg·m/s ²
f	Drag function	-
S_{ed}	Energy dissipation source term	kg/ms ⁴
E	Energy per unit mass	m ² /s ²
S_E	Energy source term	kg/ms ³
T_g	Gas phase temperature	Kelvin/ °C
$\vec{u}_g$	Gas velocity	m/s
g	Gravitational acceleration	m/s ²
q	Heat flux	W/m ²
Q_{fp}	Heat source term	J/(s·m ²)
$\vec{F}_{fp}$	Inter-phase momentum exchange term	kg·m/s
M	Mach number	-
G	Mass flux	kg/m ² s
$\vec{F}_M$	Momentum Source term	kg·m/s
N_p	Number of particles	-
d_p	Particle diameter	m
Q_{pf}	Particle fluid convection	W

Symbol	Description	Unit
m_p	Particle mass	kg
$\dot{m}_p$	Particle mass flow rate	kg/s
Re_p	Particle Reynolds number	-
$\vec{u}_p$	Particle velocity	m/s
Q_{pfp}	Particle-fluid-particle conduction	W
Q_{fp}	Particle-fluid convection	W
Q_{pp}	Particle-particle conduction	W
j_p	Particulate relaxation time	Seconds
P	Pressure	Pa
ΔP_{riser}	Pressure differential across riser	Pa
$\Delta P_{upper\ region}$	Pressure differential across upper region of the riser	Pa
$\vec{F}_p$	Pressure gradient forces	kg·m/s ²
∇p_g	Pressure gradient of gas	Pa
G_k	Source term for turbulent kinetic energy	
$C_{p,g}$	Specific heat of gas phase	J/(kg·K)
K	Thermal conductivity	W/m·K
V_p	Volume of particle	m ³
Q	Volumetric flow rate	m ³ /s
e	Wall roughness	m

CHAPTER 1 INTRODUCTION

1.1 Background and motivation

The Republic of South Africa (SA), one of the leading countries in the mining and mineral processing industry, has extensive coal resources available as fuel source to various industrial processes. The coal resources available are used primarily by coal-fired power plants for over 70% of SA's electricity generation (Morakinyo *et al.*, 2020). Because the coal used is of low grade, the burning processes release larger quantities of harmful pollutants such as carbon dioxide (CO₂), nitrogen dioxide (NO₂) and sulphur dioxide (SO₂) into the atmosphere (Morakinyo *et al.*, 2020; Shuai *et al.*, 2016). The emissions of these gaseous pollutants remains a growing problem worldwide and especially within SA as the Kriel power plant, situated in the Mpumalanga province of SA, is globally ranked as one of the most significant hotspots of SO₂ emissions from a coal combustion process (Vlavianos, 2020). These gases are recognised as priority air pollutants that must be controlled to conserve the general well-being of the environment and the public (Morakinyo *et al.*, 2020).

Measures must be taken to control these pollutant gases, specifically SO₂ and CO₂. SO₂ is a colourless, toxic, secondary greenhouse gas (GHG) which causes various environmental problems as it contributes to acid rain and localised cooling when emitted into the atmosphere (Deng *et al.*, 2019; Poullikkas, 2015). Several studies also indicate that continual exposure to SO₂ can have serious health consequences (Poullikkas, 2015). CO₂ is an acidic, colourless, primary greenhouse gas which builds up in the atmosphere and absorbs the radiant energy from the earth, resulting in a warming effect known as global warming (Sakaunnapaporn *et al.*, 2017). At low concentrations, CO₂ is harmless and even important to sustaining life, but at high concentrations CO₂ can have severe consequences for humans (Lary *et al.*, 2012). Thus, incorporating technology into existing processes is the most viable and cost-effective way to significantly mitigate SO₂ and CO₂ emissions while ensuring the relevant regulatory requirements are met.

Technologies currently in use for the abatement of SO₂ gases from coal-fired power plants are included under the umbrella of the term flue gas desulphurisation (FGD) (Wang *et al.*, 2021; Wu, Bai, *et al.*, 2020; You & Li, 2013). The absorbers used in FGD to remove SO₂ are usually calcium-based sorbents such as hydrated lime (Ca(OH)₂) (Deng *et al.*, 2019; Zhang *et al.*, 2008). FGD processes are mainly divided into three categories, namely, wet, dry, and semi-dry processes (Neathery, 1996; Wang & Zhang, 2011; Wu, Bai, *et al.*, 2020). The wet FGD process, widely used in industry, can achieve SO₂ removal rates above 95% but generates an immense amount of wet solid waste and requires the treatment of wastewater (Neathery, 1996; Wang & Zhang, 2011).

The dry FGD process is an attractive alternative compared to the wet process regarding process costs as no water treatment process is required, and the process is less energy-intensive (Chang *et al.*, 2011; Wang & Zhang, 2011). This process is, however, not widely used because of high sorbent usage, sorbent cost and low removal efficiencies (Wang & Zhang, 2011). An alternative process has been proposed from previous research, termed semi-dry FGD which can still achieve high removal efficiencies but consumes less water (Chang *et al.*, 2011; Wang *et al.*, 2021; Wang & Zhang, 2011).

The spray dry absorber (SDA) process and circulating fluidised bed flue gas desulphurisation (CFB-FGD) are examples of existing semi-dry FGD processes. The CFB-FGD process is gaining interest over SDA processes, incorporating sorbent recirculation to achieve higher SO₂ removal rates and better sorbent utilisation (Neathery, 1996). It also has lower investment costs, a simpler design, even lower water usage and no wastewater discharge (Zhang *et al.*, 2007).

Technologies available for CO₂ removal during combustion processes can be divided into three main groups, post-combustion, pre-combustion, and oxy-fuel combustion. (Abbasi & Arastoopour, 2013; Leung *et al.*, 2014). Post-combustion technology entails removing CO₂ downstream of the combustion process and requires high flue gas flow rates to reach a CO₂ concentration of above 95.5% (Leung *et al.*, 2014; Olajire, 2010). This results in large equipment, high capital costs and increased operating costs due to higher electricity requirement.

In pre-combustion technologies, flue pre-treatment is done before the combustion process. This pre-treatment entails gasification where fuel reacts with oxygen/air to produce carbon monoxide (CO) and hydrogen (H₂) (Leung *et al.*, 2014; Olajire, 2010). The CO in the mixture reacts with steam in a catalytic reactor to form CO₂ and more H₂ (Olajire, 2010). The main disadvantage of pre-combustion technology is the high capital costs associated therewith.

Lastly, oxy-fuel combustion, a modified branch of post-combustion is done in a pure oxygen environment (Leung *et al.*, 2014). Pure oxygen combustion produces flue gas mainly containing CO₂, H₂O, SO₂ and particulates. The particulates and SO₂ components can easily be removed with an electrostatic precipitator and FGD technology (Leung *et al.*, 2014; Olajire, 2010). The high concentration of CO₂ facilitates the removal thereof. The main drawback is that large amounts of oxygen are required which makes the technology expensive (Olajire, 2010).

The many advantages of CFBs play a crucial role in the efficiency of the semi-dry processes for gas removal and are therefore highly favourable for these processes (Belaid, 2017). This includes excellent heat and mass transfer rates and homogeneous reaction temperatures (Bandara *et al.*, 2019; Li *et al.*, 2014). CFBs also provide hydrodynamic heterogeneity, extended gas-particle

contact, and the higher surface area to enhance evaporation and absorption rates (Hu & Liu, 2020; Neathery, 1996). Incorporating CFB gas removal systems into existing power plants is cost-effective due to the small spatial requirements, low investment and operating costs (Deng *et al.*, 2019; Zhou *et al.*, 2011).

One of the key parameters in this desulphurisation process is humidification inside the reactor (Cui *et al.*, 2010; Gao *et al.*, 2010; Kaiser *et al.*, 2000). Humidification takes place for two main reasons. Firstly, it is to facilitate the reaction between the acid gas (SO_2) and the calcium-based sorbent and, secondly, to cool down the flue gas (Kaiser *et al.*, 2000; Wang & Zhang, 2011). In the absence of water, the reaction rate of the gas-solid reaction is low (Gao *et al.*, 2010; Wang *et al.*, 2021; Zhang *et al.*, 2007). This is also true for the reaction of CO_2 with potassium carbonate (K_2CO_3) as a sorbent, where water is essential for the reaction (Abbasi & Arastoopour, 2013; Uitbeijerse, 2011).

In addition to humidity, the temperature inside the CFB riser plays an important role in overall process efficiency. Flue gas enters the CFB at a relatively high temperature, which is reduced and becomes closer to saturation temperature by the water injected into the system (Li *et al.*, 2004; Mani *et al.*, 2008). The amount of water is adjusted to achieve the optimal operating temperature required for achieving high desulphurisation rates (Kaiser *et al.*, 2000). Temperature is also important as heat will ensure the complete evaporation of the water injected to facilitate the reaction between the acid gas and sorbent. Complete evaporation is needed to produce dry products suitable for recirculation in a CFB (Tao *et al.*, 2010a).

The design and optimisation of these semi-dry processes require one to fully understand and interpret the interaction of multiple phases inside the CFB. The interaction between these phases can be referred to as hydrodynamics, a complex branch of physics and science referring to fluids in motion and the forces acting on or exerted by the fluids (Nagy *et al.*, 2008; Prix, 2004). Several factors can influence the hydrodynamics of a fluid in a closed space, such as the CFB riser, and directly influence the pollutant gas removal efficiency (Berruti *et al.*, 1995). These factors include gas velocity, temperature, solids flux, water quantity, pollutant concentration in the flue gas and the residence time inside the CFB (Berruti *et al.*, 1995; Tritipayanon *et al.*, 2019).

Computational fluid dynamic (CFD) modelling has been proven to be an effective tool for modelling a multiphase process (Chang *et al.*, 2011; Taghipour *et al.*, 2005). CFD modelling is based on the principle of data processing and solving a set of governing equations to give a broad understanding of hydrodynamics (Dash *et al.*, 2019; Hartge *et al.*, 2009). Modelling the hydrodynamics of multiphase flow in CFB riser is typically approached by either the Eulerian-

Eulerian (E-E) or Eulerian-Lagrangian (E-L) framework (Bandara *et al.*, 2019; Benzarti *et al.*, 2012). The selected framework describes the way the different phases are treated.

The operating parameters considered in this study include inlet gas velocity, inlet gas temperature, particle mass flow rate and inlet gas relative humidity. These approaches are combined with appropriate drag, turbulence, heat transfer and evaporation models. Energy transport equations, conductivity and solid stress are a few more properties that are considered. All the properties mentioned above, and associated parameters are key to achieving a representative CFD model. Each of these has its own influence on the hydrodynamics and incorporating as much as possible into the modelling has shown more realistic results.

In this work, developing a representative CFD model assessing the influence of different operating parameters on the hydrodynamics inside a CFB riser will be attempted. In the process of developing the model, all known background and literature will be considered to achieve a credible, working model for future optimisation purposes in the semi-dry FGD and CO₂ capture processes. The methodology to execute the study entails model development, conducting experiments which resembles the modelling conditions and consequently validating the model. Finally, the modelling results, with support from experimental results will be used in discussing the hydrodynamics.

1.2 Research problem, purpose, and objectives

1.2.1 Research problem

Semi-dry flue gas treatment processes rely on numerous factors to achieve the highest possible removal efficiency. Humidity and temperature are two factors that considerably influence on the reaction rate, heat, and mass transfer of the process. The heat and mass transfer, as well as the adsorption reactions are also influenced by the hydrodynamics of gas-solid flow inside a CFB riser. However, research and information available to demonstrate the direct effect of humidity and temperature on the hydrodynamic behaviour of gas-solid flow in CFB's are extremely limited.

1.2.2 Purpose of research

The present study aims to generate and validate a computational fluid dynamics (CFD) model predicting the hydrodynamics of two suitable flue gas treatment sorbents, namely hydrated lime (Ca(OH)₂) for sulphur dioxide removal and potassium carbonate (K₂CO₃) for carbon dioxide removal at a variety of temperatures and humidities, inside a CFB riser.

1.2.3 Objectives

The following objectives are formulated:

- To determine particle properties impacting fluidisation (i.e, PSD and particle density) of hydrated lime and potassium carbonate as sorbents in CO₂ and SO₂ abatement;
- To generate a representative and validated CFD model to describe the hydrodynamics of the sorbents in a CFB riser;
- To assess the influence of two independent operating parameters, namely inlet gas relative humidity and inlet gas temperature, on the hydrodynamics inside a CFB riser, both numerically and experimentally.

1.3 Scope of study

1.3.1 Delineation of the study

The objective of this study is limited to investigating the influence of gas temperature and relative humidity on the hydrodynamics of hydrated lime and potassium carbonate in a CFB riser. This study will not include discussions based on the influence of operating parameters such as gas velocity, solids loading and changes in riser heights. This has already been addressed in previous work. Furthermore, the system is classified as a non-reacting system for the purpose of this study, and therefore, reaction kinetics will not be addressed.

1.3.2 Structure and layout of the dissertation

Chapter 1 serves as a general introduction to the study, presenting background information related to the research field and topic.

Chapter 2 commences with a literature study on general emissions and emission control strategies. The chapter then discusses the characteristics, operation, and hydrodynamics of circulating fluidised beds. Following the discussion on CFB operation, computational fluid dynamic (CFD) modelling related to emission control and circulating fluidised beds is discussed.

Chapter 3 presents the experimental methodology followed to perform the analysis and obtain experimental results, while Chapter 4 presents the methodology to develop a representative and validated CFD model to perform an analysis of the influence of temperature and humidity on the hydrodynamics of the two identified sorbents.

Model validation with experimental results and the influence of operating parameters will be presented and discussed in Chapter 5, together with a comparison of results between the two respective sorbents.

Chapter 6 of the dissertation provides concluding remarks and recommendations to consider for future work.

The dissertation structure is illustrated in Figure 1-1.

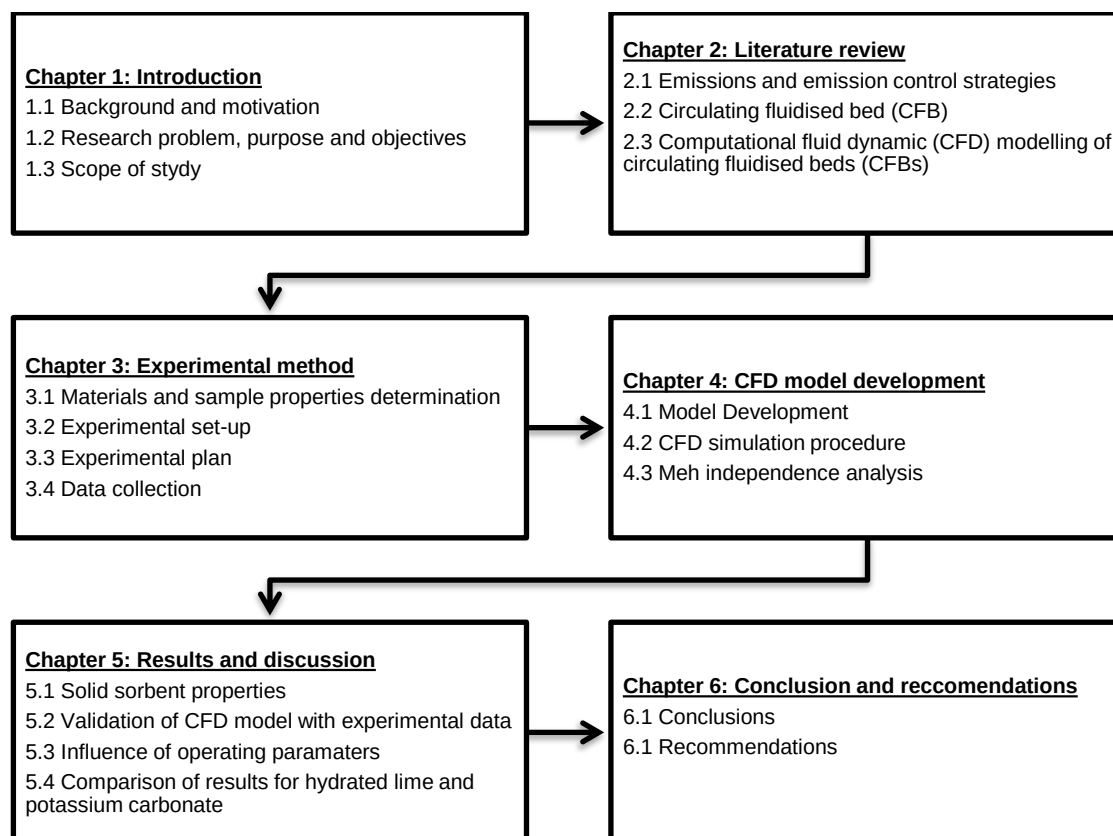


Figure 1-1: Layout of the dissertation

CHAPTER 2 LITERATURE REVIEW

This chapter introduces the current state of emissions and research into processes used for flue gas purification, outlines the working principle of CFBs and elaborates on the hydrodynamic behaviour of CFBs. It further summarises the necessary information for understanding emission control strategies, CFB's and the numerical techniques used to develop a computational fluid dynamic (CFD) model predicting the hydrodynamics in a CFB.

2.1 Emission and emission control strategies

2.1.1 The current state of emissions in South Africa (SA)

Air pollution as a consequence of industrial activities remains a growing problem in SA. In recent times, the use of fossil fuels has increased significantly to meet the growing energy demand of SA's economy (Du *et al.*, 2021; Wu, Bai, *et al.*, 2020). Coal, one of the primary fossil fuels, accounts for approximately 77% of SA's essential energy needs. The coal is primarily used as fuel for coal-fired power plants (Abbasi & Arastoopour, 2013). The ongoing combustion of these coal resources leads to the increasing emissions of largely sulphur dioxide (SO₂) and carbon dioxide (CO₂) into the atmosphere (Du *et al.*, 2021).

The emissions of SO₂ pose severe risks to human health and the environment. This air pollutant is a colourless, toxic gas, that is invisible to the human eye and has no significant odour at lower concentration (Dahiya *et al.*, 2020; Deng *et al.*, 2019; Poullikkas, 2015). It is known to be fatal when exposed to high concentrations for extended periods. One of the most damaging effects of SO₂ with respect to the environment is acid rain (Chang *et al.*, 2011; Rahimi *et al.*, 2011; Wu, Bai, *et al.*, 2020). The SO₂ molecules released into the atmosphere react with water molecules forming sulphurous acid rain (H₂SO₃), that causes damage to ecosystems and aquatic life. In addition, it causes corrosion and weathering of the infrastructure (Deng *et al.*, 2019). In 2019, SA was ranked 7th in the world for countries emitting the most significant amount of SO₂ (Dahiya *et al.*, 2020).

CO₂ is a primary GHG and the main reason for the ever-increasing problem, climate change. CO₂ absorbs the radiant energy from the earth and releases this heat gradually over time, increasing the average temperature within the earth's atmosphere (Sakaunnapaporn *et al.*, 2017). CO₂ can be harmless and even essential to life or holds serious risks depending on the concentration thereof (Lary *et al.*, 2012). SA has a large carbon footprint, and the total emissions of 2019 equated to approximately 479 Mt of CO₂. This total emissions ranked SA 12th amongst the top 20 countries emitting the most significant amount of CO₂ globally in 2019 (Dahiya *et al.*, 2020).

In SA, environmental regulations are set in place to control the amount of emissions and are published in the Government Gazette. The allowed concentration for SO₂, emitted from solid fuel combustion under normal conditions of 10% oxygen, 273 Kelvin and 101.3 kPa is 500 mg/Nm³ for newly built plants and 1000 mg/Nm³ for already established plants (Department of Environmental Affairs and Tourism, 2004). A measure to control CO₂ emissions has been implemented only recently, namely Carbon Tax (CBT). It is a method aimed to reduce GHG in a sustainable, cost-effective and affordable manner.

Despite the fact that the world is moving towards increased sustainability via improved regulatory action, the power plants within SA are still not complying with stringent environmental regulations. It is therefore of imperative importance that measures and technology must be implemented to lower the occurrence of emissions and better control the impact thereof.

2.1.2 Flue gas desulphurisation (FGD)

Continuous industrial development world wide contributing to increased SO₂ emissions has led to the development of processes and strategies to effectively reduce SO₂ emissions (Wang *et al.*, 2021). FGD is the most implemented and well-known process for removing SO₂ from waste flue gas streams. Many researchers also recognize FGD as the most effective process for this application (You & Li, 2013). FGD can be described as a process whereby sorbents chemically react with the SO₂ in an adsorption reaction. The sorbent, usually calcium-based, is chosen based on the relevant process requirements, availability and economic grounds. The most commonly used sorbent is hydrated lime (Ca(OH)₂) (Pisani & De Moraes, 2004). Furthermore, existing FGD processes can be classified into three primary groups based on the involvement of water in the process, namely dry, semi-dry and wet FGD processes (Chang *et al.*, 2011; Mohanty *et al.*, 2009).

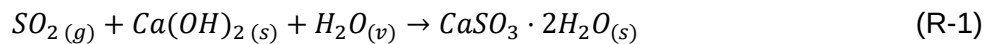
Semi-dry FGD processes have been developed to combine the advantages and eliminate the disadvantages of both wet and dry FGD processes (Chang *et al.*, 2011; Wang *et al.*, 2021). The desired advantages of semi-dry FGD processes include a rapid reaction rate, high desulphurisation efficiency, lower water consumption, lower spacial requirements and lower capital cost (Chang *et al.*, 2011; Neathery, 1996). In addition, since the energy dries the sorbent in the flue gas, the semi-dry FGD processes do not require costly dewatering equipment (Neathery, 1996).

Two traditional semi-dry FGD processes are spray-drying adsorption (SDA) and circulating fluidised bed flue gas desulphurisation (CFB-FGD) (Chang *et al.*, 2011; Hill & Zank, 2000). Both processes have been used widely in the utility industry for their ability to remove multiple pollutants at once (Deng *et al.*, 2019; Koech *et al.*, 2021). Spray dry adsorption entails mixing

Ca(OH)_2 with excess water to produce a slurry that reacts with the SO_2 in the flue gas (Hill & Zank, 2000; Koech *et al.*, 2021). For the total desulphurisation reaction to occur, as well as complete slurry evaporation, the gas residence time in spray dry adsorption is typically between 12-30 seconds (Chang *et al.*, 2011; Koech *et al.*, 2021; Zhang & Wang, 2012).

CFB-FGD employs all the advantages of CFB technology (detailed in Section 2.2) to enhance the overall SO_2 capture efficiency. The process configuration, unlike the case of SDA, allows for a variation of methods to add water to the process (Deng *et al.*, 2019; Gao *et al.*, 2010; Koech *et al.*, 2021). The recirculation of sorbent results in prolonged contact time between the flue gas and sorbent and subsequently, better sorbent utilisation (Grace & Lim, 2013; Zhang *et al.*, 2008). CFBs demonstrated better fuel flexibility over spray dry absorbers, which allows for a broader range of fuels' sulphur content (Fischer & Darling, 2015). For these reasons, CFB-FGD is regularly chosen to retrofit existing power plants due to its inherent versatility and robustness. Removal efficiencies of up to 98% can be achieved with the correct operation and process conditions (Li *et al.*, 2011).

Humidification is a crucial precondition and key variable in semi-dry FGD processes (Cui *et al.*, 2010; Gao *et al.*, 2010). It is necessary for achieving a high desulphurisation efficiency by serving two main purposes, namely (i) to facilitate the reaction between Ca(OH)_2 and the SO_2 present by transition of the gas-solid reaction to an ionic reaction; and (ii) to lower the incoming flue gas temperature (Wang *et al.*, 2021; Yoon *et al.*, 1988; Zhang *et al.*, 2007). It is known that ionic reactions are instantaneous in the aqueous phase, and therefore humidity is essential to the reaction rate (Zhang *et al.*, 2007). The overall reaction between Ca(OH)_2 and SO_2 in a humid environment can be given as:



Humidification is known to significantly influence the desulphurisation process organisation, utilisation of the hydrated lime and wall deposition within the reactor (Cui *et al.*, 2010; Gao *et al.*, 2010). For this reason it is important that the humidification mechanism is developed and designed carefully to improve the overall process efficiency.

Various authors have identified the three mechanisms that exist for achieving the desired level of humidity including Cui *et al.*, 2010; Moyeda *et al.*, 2002; Wang & Zhang, 2011 as:

1. Pre-slurrying of hydrated lime;
2. Dry sorbent injection with separate spraying of water into the reactor;
3. Pre-humidification of the flue gas.

The pre-slurrying of hydrated lime requires a slurry make-up and handling system, which increases the investment cost and system spacial requirement. Therefore the other two mechanisms are applied more often in semi-dry CFB-FGD processes (Chang *et al.*, 2011). The other semi-dry FGD processes consist of a configuration pertaining to dry sorbent injection and the addition of humidity by means of separate water injection or pre-humidification of the flue gas (Koech *et al.*, 2021; Moyeda *et al.*, 2002). The configuration of the respective humidification mechanisms differs slightly.

The separate injection of water in the form of fine droplets involves a few important considerations for efficient operation. The water is normally added by means of a nozzle, spraying fine droplets into the reactor (Gao *et al.*, 2010). This fine spray can either be co-current or countercurrent in relation to the flue gas flow direction. The droplet diameter is one of the important considerations and is directly related to the type of nozzle used. Smaller droplets lead to an overall higher surface area than large droplets and therefore enhances adsorption rates (Du *et al.*, 2021; Neathery, 1996). However, smaller droplets have a higher evaporation rate; thus, their effective wet lifetime is shorter than that of larger droplets (Neathery, 1996). Another important factor, especially within an enclosed space, is the spray angle that is formed by the water discharge of the nozzle (Tao *et al.*, 2010b). Whenever water is introduced as a separate component, wall deposition is present to a certain degree (Cui *et al.*, 2010; Gao *et al.*, 2010). To decrease the degree of wall deposition, the spray angle should be chosen to benefit efficient wetting of sorbent particles and to minimise wall wetting. The water flow rate is a third factor to take into consideration, as over-wetting of the particles can easily occur (Gao *et al.*, 2010). This mechanism is complex and difficult to successfully accomplish without compromising on some key considerations such as removal efficiency.

Depending on the system, single and multi-stage humidification configurations can be used to add water to the system (Cui *et al.*, 2010; Gao *et al.*, 2010). Single humidification configurations are typically used in CFB's but have difficulty achieving high gas removal efficiencies while decreasing the deposition of water droplets on the reactor wall (Cui *et al.*, 2010; Shuai *et al.*, 2016; Uitbeijer, 2011). Scaling due to over-wetting of particles is also observed in single humidification (Cui *et al.*, 2010; Gao *et al.*, 2010). The quantity of water added can be lowered to avoid over-wetting but will decrease removal efficiency. Multi-stage humidification is a method developed to distribute the water more uniformly across the reaction field (Cui *et al.*, 2010; Gao *et al.*, 2010). This method is used to keep the humidity content of the sorbent above the critical moisture point for an extended period. It is also observed that scaling on the wall of the reactor can be reduced with multi-stage humidification (Gao *et al.*, 2010).

Pre-humidification of the flue gas is a third mechanism for achieving humid conditions in the FGD process (Moyeda *et al.*, 2002). Water in the form of a fine spray is introduced into the flue gas, downstream from the air-preheater and upstream of the sorbent injection point. The water spray is almost immediately evaporated into a water vapour by the temperature of the flue gas (Moyeda *et al.*, 2002). The level of humidity can be pre-determined and reached by adjusting the amount of water sprayed through the nozzles. This method of creating a humid environment is usually employed to reduce scaling and wall depositions in the reactor (Cui *et al.*, 2010; Zhang *et al.*, 2007). It has been reported that a SO₂ capture efficiency of 60% can be achieved with pre-humidification of the flue gas when using Ca(OH)₂ as the sorbent (Córdoba, 2015). Furthermore, seeing that water is still a component injected separately, the considerations discussed above for the method of separate injection of water directly into the process reactor is still applicable and important to take into account (Du *et al.*, 2021; Gao *et al.*, 2010; Tao *et al.*, 2010b).

2.1.3 Carbon capture and storage (CCS)

To mitigate CO₂ emissions, researchers (Nord & Bolland, 2020) proposed four distinct principles which can be applied:

1. Decrease deforestation activities and increase afforestation;
2. Consume lower levels of fossil fuels by using alternative energy sources and reducing the need for energy;
3. Use of fossil fuels with a lower carbon-to-hydrogen ratio;
4. Carbon capture and storage (CCS) technologies.

Acknowledging that economic growth occurs exponentially and directly leads to increases in energy demand, studies concluded that achieving a significant reduction in CO₂ emissions will likely be impossible without implementing CCS technologies.

CCS technologies are used to separate CO₂ from a source point, transport the CO₂ to a storage location and isolate the captured CO₂ from the atmosphere for a long period of time. Implementing CCS technologies in coal-fired power plants can accept one of three approaches (Abbasi & Arastoopour, 2013; Leung *et al.*, 2014):

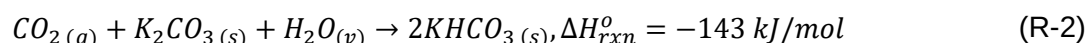
1. Pre-combustion
2. Post-combustion
3. Oxy-fuel combustion

Currently post-combustion processes are considered very mature and effective in capturing CO₂ from coal-fired power plants. These processes entail the addition of CCS technology downstream of the combustion process without significantly influencing the power generation process (Guo *et al.*

al., 2021; Leung *et al.*, 2014). Depending on the process requirements and specifications, post-combustion processes include absorption, adsorption, cryogenic separation and membrane separation (Guo *et al.*, 2021; Shuai *et al.*, 2016).

Adsorption post-combustion processes utilize a solid sorbent to capture CO₂ from the flue gas stream. The CO₂ is separated from the flue gas by means of a chemical reaction and binds to the surface of a porous sorbent. The type of sorbent used is of great importance and chosen based on certain factors, including adsorption capacity, CO₂ selectivity, process conditions and the cost and availability of the respective sorbent (Guo *et al.*, 2021). Sorbents typically used for post-combustion adsorption processes include activated carbons, zeolites, polymer- and silica-based sorbents and alkali-metal carbonates. Potassium carbonate (K₂CO₃) is chosen frequently because of its excellent CO₂ adsorption capacity, low decomposition pressure and high-level attrition resistance (Abbasi & Arastoopour, 2013; Chalermssinsuwan *et al.*, 2010).

The interaction between K₂CO₃ and the flue gas is very important and influences the removal percentage of CO₂ for the reason that K₂CO₃ captures CO₂ via the following chemical reaction (Borhani *et al.*, 2015; Shuai *et al.*, 2016):



This reaction is exothermic; therefore, no additional heat will be necessary to control the reaction temperature (Chalermssinsuwan *et al.*, 2010). In certain cases, a medium will be necessary to remove some of the excess heat from the reaction zone.

The water required for the adsorption reaction is responsible for cooling down the incoming flue gas and activating the K₂CO₃ to improve the reaction for CO₂ removal. The water is introduced to the process by means of pre-humidification of the incoming flue gas which typically increases the relative humidity levels to range between 7%-30% (Yi *et al.*, 2007). The level of humidity significantly influences the CO₂ removal efficiency of the process. Studies have found that the adsorption reaction removes high percentages of CO₂ at lower temperatures more effectively (Guo *et al.*, 2021). However, this can be problematic as condensation of the water vapour present in the flue gas has been noticed as the temperature decreases. K₂CO₃, being a hygroscopic salt, would capture all of the condensed water and cause particle aggregation to occur. The aggregation will directly influence the hydrodynamic behaviour witnessed in the reactor and process parameters such as pressure drop across the bed (Guo *et al.*, 2021). It is evident that the efficient operation of the process is directly related to the control of both the humidity level and temperature to optimal levels.

2.1.4 The importance and review of research towards emission control strategies

Increasing environmental awareness and movement towards a more sustainable future has led to extensive research on mitigation strategies and control of SO₂ and CO₂ emissions. The growing levels of anthropogenic SO₂ and CO₂ in the atmosphere from the combustion of fossil fuels are considered the primary contributors to global warming (Morakinyo *et al.*, 2020). Consequently, various processes to be implemented, pre- or post-combustion, have been introduced and investigated to reduce emissions significantly.

Pre-combustion processes entail the removal of undesired chemicals from the fuel source before combustion. In contrast, post-combustion processes entail the removal of undesired compounds from the resulting flue gas (Miller, 2011). Currently, research focuses primarily on removing SO₂ and CO₂ from flue gases since the complete removal of undesired chemicals from the fuel source is not always possible in pre-combustion processes (Deng *et al.*, 2019). In addition, these processes are currently not technically or economically viable (Deng *et al.*, 2019; Yalcin *et al.*, 2020).

Post-combustion carbon capture and storage technologies are the most preferred and commercially applied technologies for the removal of CO₂ from flue gasses (Yalcin *et al.*, 2020). Although some technologies are viable and applied commercially, absorption-based technology is most commonly studied and used. One major drawback of these technologies is the economic losses incurred when lower concentrations of CO₂ are present in flue gasses (Leung *et al.*, 2014; Yalcin *et al.*, 2020).

In the case of SO₂ removal, the post-combustion wet and semi-dry processes are predominantly applied in industry (Srivastava & Jozewicz, 2001; Zhao & Zhang, 2021). Research is mainly concerned with improving these technologies in terms of decreased water consumption, increased gas-sorbent contact efficiency and increased SO₂ removal efficiency (Deng *et al.*, 2019; Srivastava *et al.*, 2001; Srivastava & Jozewicz, 2001).

The research towards emission control strategies has developed significantly, but major challenges are still present and need to be solved. As a result of the extensive research primarily towards continual improvement of conventional strategies, a few new technologies have emerged and been proposed. However, with developing new technologies, researchers face many challenges such as high energy requirements, the need and availability for new materials, high resource consumption rates, high investment, and operational costs (Leung *et al.*, 2014; Yalcin *et al.*, 2020). Therefore, a substantial amount of developmental work is required to mature and establish these technologies (Yalcin *et al.*, 2020). Research at present mainly focusses on

improving removal efficiencies, reducing operational requirements and costs, improving the quality of by-products, and reducing secondary emissions (Deng *et al.*, 2019; Leung *et al.*, 2014).

Most of the technologies employed to control emissions, rely on reactions between a chemically active sorbent and unwanted components in the flue gas (Deng *et al.*, 2019). As a result of these methods, multiple phases (solids, gases, liquids) will always be present in the process to achieve the highest possible removal efficiency. Multi-phase flow is a large driving force behind the ongoing research towards mitigation strategies because of the complex flow behaviour (Hodapp *et al.*, 2012). In addition, the nature of these technologies involves complex reactions and flow behaviour of the respective phases, which are also not completely understood especially in terms of their interaction which directly influences reaction rates, heat and mass transfer and the overall removal efficiencies (Deng *et al.*, 2019). Therefore, a great amount of research effort is still required to establish a complete understanding of the flow behaviour and underlying reactions occurring in these technology systems. This will provide researchers with the potential to improve the economic and technical viability of these technologies in order to be more effectively employed as a means of emission control.

As mentioned above, an increasing demand for power generation exists due to global technological development and fast population growth (Du *et al.*, 2021). This indicates that coal-fired power plants will remain part of the global energy supply system for the foreseeable future. The further development of currently viable and effective technologies as well as the development of new, more efficient technologies and investigations into the application of hybrid technologies, is imperative for future emission control (Yalcin *et al.*, 2020).

2.2 Circulating fluidised beds (CFB)

2.2.1 Characteristics and operating conditions of CFBs

In the past few decades, circulating fluidised beds (CFBs) have been studied extensively within the field of multiphase flow and fluidisation. The studies performed aimed to continuously improve processes involving CFBs by better understanding of the multiphase flow phenomena and all related concepts. The complication, however, is that with multiphase flow, the phases can be dispersed radially and axially across the CFB riser, adopting a chaotic flow behaviour that drastically influences the design and operation of CFB technology (Hodapp *et al.*, 2012).

Given its numerous advantages, CFBs have replaced equipment traditionally used for processes involving gas-solid and gas-catalytic reactions (Belaid, 2017). These include large volumetric throughput, high fuel flexibility, excellent gas-solid mixing, significant mass and heat transfer

rates, temperature uniformity and extended gas-solid contact time (Horio, 2013; Panday *et al.*, 2016; Tu *et al.*, 2022).

As the name suggests, a CFB's working principle is based on fluidisation, a fundamental and powerful method in the engineering industry to handle various solid materials (Ghadirian *et al.*, 2019; Wang & Zhu, 2016). Fluidisation in CFBs can be termed a process in which a load of solid particles enters the riser at the bottom and adopts a fluid-like behaviour as a result of the upward-flowing gas (Belaid, 2017; Cocco *et al.*, 2014). The gas imposes a large enough drag force on the solid particles to eliminate the downward forces present (Cocco *et al.*, 2014). The particle pathway is further influenced by fluid-mechanical, collisional and surface forces (Horio, 2013). The most important parameter of fluidisation is the minimum fluidisation velocity (u_{mf}). This is the superficial gas velocity (u_g) required to counteract the gravitational force on the particle by the drag force imposed on the particle by the upward flowing gas. As the superficial velocity of the gas increases above the minimum fluidisation velocity, different regimes can be observed such as bubbling, turbulent, fast and conveying fluidisation.

A conventional CFB system consists of three functional components; a riser for gas-solid interactions, a cyclone for separating the phases and a return leg for solid recirculation (Belaid, 2017; Panday *et al.*, 2016). In most systems, solid particles and gas enter the riser from the bottom section which causes unsteady flow and contributes to the complexity of the flow inside the riser (Chang *et al.*, 2011). The riser is the key component of a CFB system, as all the process reactions occur within this space. For this reason, it is important to ensure that a suitable flow field, although a bit unsteady, should be maintained to minimise scaling and abrasions while maximising the reaction rates (Gao *et al.*, 2010). The layout of a typical CFB system is depicted in Figure 2-1 below.

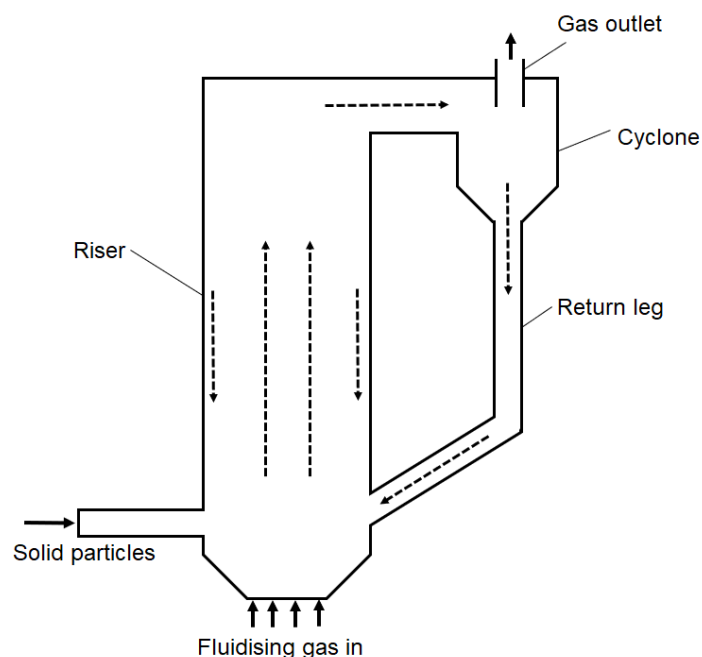


Figure 2-1: Schematic representation of a typical CFB configuration

Adapted from (Grace *et al.*, 2003)

The operation and effectiveness of CFBs rely on the operating conditions, riser geometry, hydrodynamics and heat and mass transfer rates (Grace & Lim, 2013). The operating conditions are highly dependant on the process and specific requirements. For commercial operation, the dominant operating conditions can be specified to fall within certain ranges. These ranges are only a general indication, and exceptions to specific processes do exist. The parameters and operating ranges are summarised in Table 2-1. (Berruti *et al.*, 1995; Patience *et al.*, 1992).

Table 2-1: Commercial operating parameter ranges for CFB's

Parameter	Operating range
Superficial gas velocity	2-12 m/s
Solids Flux	10 -1000 kg/m ²
Pressure	100 – 2000 kPa
Temperature	20-950 °C
Mean particle diameter	50-500 µm
Total riser length	15-40 m

2.2.2 Hydrodynamics in circulating fluidised beds (CFB's)

It is well-established that circulating fluidised bed (CFB) technology is the preferred choice of equipment in numerous industrial processes. The previously mentioned advantages make it suitable for an extensive range of applications within the engineering industry. However, for defining a CFB's performance, as well as improving the operating conditions and process efficiency, the hydrodynamics in CFBs are extremely important. For this reason, and due to the complexity and diversity of multi-phase flows in an enclosed space, it is important that research related to the hydrodynamics of multi-phase flow in CFBs with the aim of continuous improvement in the field remains a growing topic.

The intensive research put into CFB hydrodynamics in the last few decades has led to a sufficient understanding of several aspects of multiphase flow behaviour in risers. However, some important areas are less well-understood with limited information available in the literature. The rapid advances in the engineering industry require progressively accurate predictions and data on the hydrodynamics of multiphase flow systems. Therefore, it is necessary to use well-known information available to elucidate the unknown areas. The currently available research studies can be divided into two primary groups; full-loop studies and riser hydrodynamic studies (Tu *et al.*, 2022). Researchers focus more on only the riser section, being the process reactor in a CFB system. Hydrodynamics studies generally focus on flow behaviour including, solids hold-up, particle size distribution (PSD), velocity distribution, pressure drop and cluster formation (Yang & Wang, 2020).

Hydrodynamics in the riser are directly related to multiphase flow as phenomena and fluidisation theory. In the case of gas-solid flow in the riser, the solid phase properties such as particle diameter and density influences flow patterns, minimum fluidisation velocity, mass transfer and available contacting surface area. The gas phase is responsible for carrying the solid phase; therefore, the magnitude of the gas velocity is important to achieve fluidisation. The gas velocity is also responsible for the transitions between the different fluidisation regimes, and therefore, a desired fluidisation regime can be achieved by proper control of the gas velocity (Philippsen *et al.*, 2015). Furthermore, the hydrodynamics observed during experimentation and modelling, depend on the operating conditions of the system. However, certain observations are universal, and some definite conclusions have been made regarding the hydrodynamics in CFBs.

In CFB systems, the quantitative importance of the hydrodynamic behaviour of the solid evident in the fact that the rate of both heat and mass transfer is highly influenced by different hydrodynamic regimes (Hodapp *et al.*, 2012; Sakaunnapaporn *et al.*, 2017). Experimental investigations show that during operation, the CFB riser demonstrates an inhomogeneous axial

solids hold-up distribution with some degree of densification at the bottom of the riser where solids are normally introduced (Berruti *et al.*, 1995; Hussain *et al.*, 2005; Varas *et al.*, 2017). The solids are accelerated along the height of the riser by the incoming gas, and therefore, the solids hold-up decreases along the height of the riser (Berruti *et al.*, 1995; Neri & Gidaspow, 2000). Although the axial solid distribution is inhomogeneous, it is still a relatively important observation to be made as it determines the pressure drop along the riser to a great extent (Hodapp *et al.*, 2012; Hussain *et al.*, 2005).

In the radial direction, an inhomogeneous solids hold-up distribution can also be observed, but this observation is inadequately understood by researchers (Helland *et al.*, 2007). It may be the reason for the significant downflow of solid particles near the riser wall observed during research studies. The interphase and inter-particle interactions in the riser cause particles to move in a fluctuating motion and contribute to the inhomogeneous distribution radially (Yang & Wang, 2020). Other reasons identified include, but are not limited to, the riser geometry, and solid and gas-phase inlet configuration. The inlet-outlet configuration can also develop a strong asymmetric flow profile in the CFB riser (Berruti *et al.*, 1995; Yang & Wang, 2020). The particle residence time distribution (RTD) is largely influenced by the inhomogeneous solids hold-up distribution in a CFB riser (Hodapp *et al.*, 2012; Niewland *et al.*, 1994).

It is frequently seen in research studies that hydrodynamics in a riser are characterised by co-current gas-solid flow in the core of the riser and solids downflow near the riser wall (Hiidenkari, 2018; Neri & Gidaspow, 2000; Varas *et al.*, 2017). This is the so-called core-annulus flow structure and can be seen in Figure 2-2. This type of flow structure is usually observed at operating conditions where the average upward flowing velocity is between 2 m/s and 8 m/s and the average upward flux of solids is 15-90 kg/m²·s (Davidson, 2000; Neri & Gidaspow, 2000). However, the parameters can slightly differ depending on the application, operating scale and combination of operating conditions such as mass flux, gas velocity and particle diameter (Ding *et al.*, 2008; Sanaei *et al.*, 2010). Clusters are continually formed and disintegrated by the upward and downward motion and tend to be more concentrated in the downflow region near the riser wall (Cahyadi *et al.*, 2017; Hussain *et al.*, 2005). As the flow becomes more developed along the height of the riser, the solid back mixing gradually decreases (Hussain *et al.*, 2005; Yang & Wang, 2020).

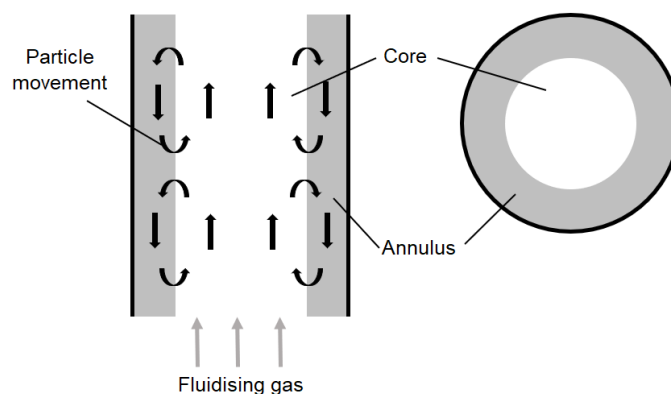


Figure 2-2 Schematic of core-annulus flow pattern in CFB's

Adapted from (Hiidenkari, 2018)

Some researchers have acknowledged the effect of humidity on the flow behaviour inside CFB risers during investigations. However, the understanding of the flow behaviour of gas and solid particles in a moist environment remains limited and needs some further investigations (Wang *et al.*, 2021). During a practical and numerical investigation by Bing Du, 2005, it was determined that the relative humidity (RH) of the fluidising gas influences the electrostatic forces of particle-particle and particle-wall interactions. The change in the strength of electrostatic forces can affect the solids hold-up in the radial direction. However, the literature has not formally addressed this phenomenon (Estrella, 2021).

Studies investigating temperature as a parameter influencing the hydrodynamics are still limited and poorly developed. A few important observations have, however, already been made. Sanaei *et al.*, 2010 reported that with a temperature increase from ambient conditions up to around 300°C, the axial solids mixing and particle diffusivity also tend to increase. At temperatures above 300°C, the mixing has shown a decrease. It was also found that temperature inside the CFB affects the gas viscosity and gas density which influence the interaction between the gas and particles (Ding *et al.*, 2008; Yates, 1996; Zhang *et al.*, 2018). Observations made with an increase in gas viscosity include a slight increase in the solids hold-up in the bottom region of the riser (Zhang *et al.*, 2018). Radially, the solids concentration decreases slightly and axial solids velocity increases in the core zone of the riser, with an increase in temperature (Sanaei *et al.*, 2010; Zhang *et al.*, 2018). For the fluidisation of fine particles, the inter-particle forces are considerably influenced by the temperature inside the riser (Cui *et al.*, 2003; Hodapp *et al.*, 2012). According to the Wen-Yu fluidisation correlation, gas viscosity is the dominant variable when fluidising small particles, while the effect of density is more pertinent for larger particles (Amblard, 2020).

2.2.3 Application of CFBs in emission control technology

CFB technology is applied as a means of flue gas treatment in several industries, including but not limited to municipal solid waste (MSW) incineration systems, cement plants, glass works, metallurgical plants and power production plants (Yi *et al.*, 2005). At present there are numerous CFB technology plants in operation world-wide that serve as a means of removing primarily SO₂ and CO₂ from flue gasses (Cai *et al.*, 2017; Leuschke & Babcock, 2008; Yi *et al.*, 2005). Table 2-2 lists some of the known CFB applications in different industries.

Table 2-2: Reported commercial CFB application worldwide

Location	Industry	Application	References
China	Power generation	CFB semi-dry desulphurisation process downstream of two 300 MW power generating units.	(Zhao & Zhang, 2021)
Ireland	Power generation	CFB dry desulphurisation process downstream of three 300 MW power plants.	(Yi <i>et al.</i> , 2005)
Austria	Metallurgical plant	CFB dry desulphurisation process downstream of a rotary kiln for regeneration of catalysts. SO ₂ removal efficiency of ≥ 99.7%.	(Yi <i>et al.</i> , 2005)
Germany	MSW incineration	CFB dry desulphurisation process downstream of an MSW incineration process. SO ₂ removal efficiency of ≥ 93.3%.	(Yi <i>et al.</i> , 2005)
Spain	MSW incineration	CFB dry desulphurisation process downstream of an MSW incineration process. SO ₂ removal efficiency of ≥ 92%.	(Yi <i>et al.</i> , 2005)

From Table 2-2 the dominance of CFB technology in abatement strategies for air pollution is evident in the power generation industry as well as other flue gas emitting industries.

2.2.4 Evaluation of research towards CFB hydrodynamics

The study of gas-solid hydrodynamics in circulating fluidised beds (CFBs) plays a significant role in defining the beds' performance. The complexity of the system gives researchers many more topics to investigate in order to contribute to the clarification and understanding of CFB hydrodynamics. It is noticed that limited studies are available where the direct influence of temperature and/or humidity on the hydrodynamics are investigated. Table 2-3 summarises a few studies where temperature and/or humidity influences on flow behaviour were mentioned along with relevant results obtained in these particular studies.

Table 2-3: CFB hydrodynamic studies where temperature and/or humidity is investigated

Reference	Conditions	Measured/ Analysed	Experimental/ Numerical	If numerical, which model?	Findings
(Ding <i>et al.</i> , 2008)	1. T=19°C; 100°C 2. Superficial gas velocity (U_g) & solids mass flux (G_s) varied	1. Pressure drop across riser (ΔP) 2. Total solids hold-up	Both	Two fluid model (E-E)	1. For a specific solids load and superficial gas velocity, ΔP is higher at 100°C than at 19°C. 2. Total solids hold-up is higher at 19°C than at 100°C, at each respective U_g .
(Zhang <i>et al.</i> , 2018)	1. T=26.85°C, 326.85°C; 926.85°C (gas viscosity = 1.8×10^{-5} Pa·s; 3.03×10^{-5} Pa·s; 4.64×10^{-5} Pa·s)	1. Solids volume fraction (radial & axial) 2. Solids axial velocity in radial profile	Numerical	Two fluid model (E-E)	1. Increase in temperature causes an increase in gas viscosity. 2. Solids volume fraction increase slightly in bottom of riser as gas viscosity increases. 3. Core zone: solids axial velocity rises and solids concentration slightly decreases with increasing gas viscosity. Solids tend to accumulate at wall regions.
(Bing Du, 2005)	1. RH = 6.5%; 20%; 40%.	1. Solids concentration in core zone of CFB. 2. Solids-hold up	Both	Not specified	1. RH increases from 6.5% to 20%: area occupied by solids core region decrease a. Electrostatic forces of p-p and p-w decrease with increasing RH 2. RH increases from 20% to 40%: area occupied by solids core region increase

Reference	Conditions	Measured/ Analysed	Experimental/ Numerical	If numerical, which model?	Findings
					a. Electrostatic forces of particle-particle and particle-wall further decreased b. Enhanced exchange of solids in radial direction (enlarged solids core region) 3. Difference in solids hold-up of lower and upper part is decreases (more uniform in axial direction)

2.3 Computational fluid dynamic (CFD) modelling of circulation fluidised beds (CFB's)

2.3.1 Introduction

When investigating the gas-solid flow behaviour in circulating fluidised beds (CFBs), there are normally two approaches, i.e., experimental measurements and computational fluid dynamic (CFD) modelling. The flow structure in CFBs is inherently complex and vary in time and space domains (Panday *et al.*, 2016). Experimental measurements are generally macro-scale observations; therefore, additional techniques must be employed in gas-solid flow behaviour studies of CFBs.

With the extensive computational power available and the advanced modelling methods defined, CFD has become an effective tool for modelling the flow behaviour in CFBs and holds great potential for long-term usefulness. CFD has advanced from two-dimensional (2-D) to three-dimensional (3-D) models and application ranges from laboratory-scale systems up to commercial-scale systems (Tu *et al.*, 2022). CFD simulations can provide detailed insight into complex flow behaviour, without extensive experimental work or flow disturbances typically encountered during experimental measurements (Zhang *et al.*, 2012).

As a result of computational power limitations, CFD studies have been limited to investigating isolated parts of CFBs (Tu *et al.*, 2022). Risers are the most common part of interest since it is where the process reactions take place. The only downfall of isolated modelling approaches is the specification of boundary conditions and the inability to capture the dynamic response of the CFB system as a whole when conditions are changed (Li *et al.*, 2014). However, recently full-loop hydrodynamic modelling is being investigated more often and has shown exceptional improvement in the recent years.

General CFD modelling studies conducted by researchers on flow behaviour inside CFBs include the effect of operating parameters on CFB flow behaviour, the effect of different drag models, riser geometry influence, etc. Numerical methods, inter-phase drag models and turbulence models are the main factors included in CFB simulations.

2.3.2 Numerical methods for modelling CFB hydrodynamics

The numerical approaches available to model and describe CFB multiphase hydrodynamics can be classified according to the framework in which the dispersed and continuum phases are modelled (Tu *et al.*, 2022). In gas-solid flows, the gas phase is treated as a continuum, and the solid particles as the dispersed phase. The Eulerian-Eulerian (E-E) and Eulerian-Lagrangian (E-L) are two traditional approaches to classify typical models used in literature. The E-E approach

treats both the gas and solid phases as fully interpenetrating, continuous phases (Ariyaratne *et al.*, 2018; Bandara *et al.*, 2019). A set of equations for continuity, momentum and turbulence are solved for each phase. This approach is suitable for high particle concentration systems, where interaction and motion occurs. The E-E approach is often coupled with the kinetic theory of granular flow (KTGF) to account for particle-particle interactions. The E-E approach is more readily used by researchers as it requires less computational strength but has the disadvantage of not being able to model particle size distributions (Chen & Wang, 2014).

In contrast, the E-L approach treats the solid phase differently from the E-E approach by tracking the pathway of each particle or a group of particles with the same characteristics (Ariyaratne *et al.*, 2018; Tu *et al.*, 2022). In the E-L approach the gas phase mass and momentum transfer are described by the Navier-Stokes time-averaged equations, and the solid phase is described by the Newtonian equations of motion to track the movement of each particle or particle group. There exist four distinct models developed according to the E-L approach, but each is different in the way which particle-particle (p-p) interactions are treated. These models are Discrete Phase Model (DPM), Dense Discrete Phase Model coupled with the KTGF (DDPM-KTGF), CFD-Discrete Element Model (CFD-DEM) and lastly, MultiPhase-Particle-In-Cell (MP-PIC). These models have the advantage of handling various particle sizes, densities, and velocities. The drawback however is that modelling according to the E-L approach requires that the number of particles should be considered ($<10^6$). In Figure 2-3, a general classification of typical models used to study gas-solid hydrodynamics is depicted.

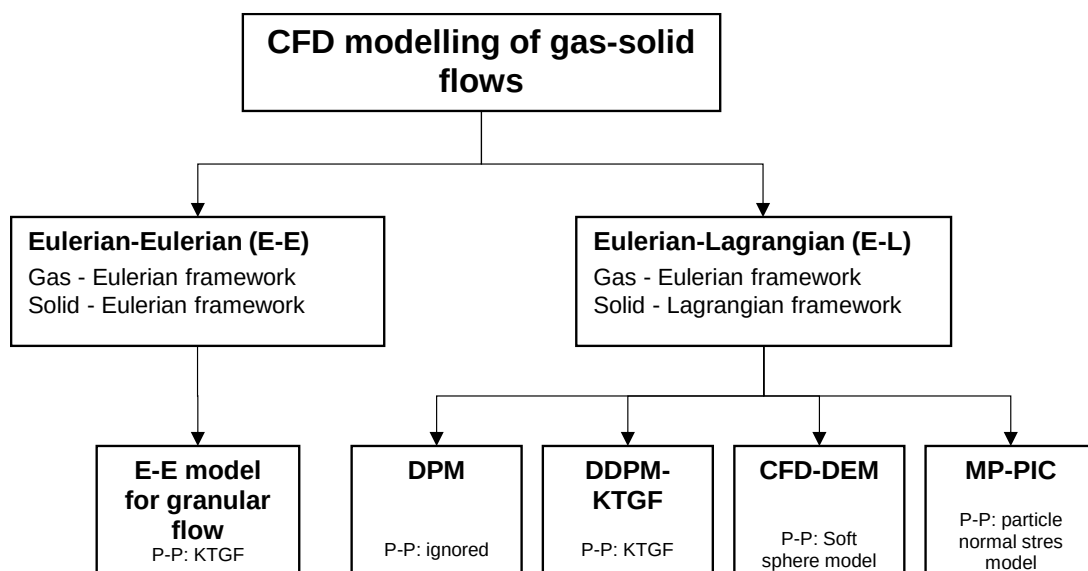


Figure 2-3: CFD model classification

2.3.3 Review of recent Eulerian-Lagrangian (E-L) and Eulerian-Eulerian (E-E) approach on CFB hydrodynamic studies

In Table 2-4, some recent studies with an Eulerian-Lagrangian (E-L) approach or an Eulerian-Eulerian (E-E) approach are given. In the Table some of the highlighted aspects of the studies are summarised such as the associated drag model and turbulence model used, together with additional specifications. The main findings are also summarised. From the studies summarised in the table, it can be concluded that homogeneous drag models are used widely considering the lower computational cost reported and ability to predict reasonable flow results in combination with the correct modelling parameters. However, it is also reported that a heterogeneous drag model, such as the EMMS-drag model significantly improved the prediction of gas-solid flows, but requires an immense amount of simulation time and sometimes geometry simplifications (Zhang & You, 2016; Zhu *et al.*, 2020). The approach used for modelling is primarily based on the purpose of the study. In studies where parameters such as solids distributions and solids velocities would be some of the main objectives, the E-L approach is preferred in its ability to describe particle trajectories and handling a wide range of particle properties, however, with tracking the individual particles comes the requirement of large computational cost and simulation time. Usually the E-L approach is used in more dilute systems.

Table 2-4: Recent E-L and E-E CFB hydrodynamic studies

Reference	Model	Drag model	Turbulence model	Other specifications	Main Findings
(Yang & Wang, 2020)	MP-PIC	Combined Wen-Yu & Ergun model	Not specified	Study to evaluate gas-solid characteristics and transport mechanisms	- Model had good agreement with experimental data. - Identified core-annulus flow from the model.
(Yin <i>et al.</i> , 2014)	MP-PIC	Gidaspow model	LES	Sensitivity of drag model and other modelling parameters tested and verified	- Gidaspow drag model shows better axial pressure drop profile and radial particle volume fraction distributions. - Non-ideal particle-particle interaction gives more reasonable results than perfect elastic particle collision ($e_p=1.0$)
(Zhang <i>et al.</i> , 2012)	CPFD	Combined Wen-Yu & Ergun model	Algebraic	Various superficial gas velocities and solids mass fluxes in tow regimes: 1. Dilute phase transport 2. Fast fluidisation	- CPFD modelled main features well in both regimes. - Clustering and solids back mixing were stronger in FF regime.
(Alobaid <i>et al.</i> , 2013)	CFD-DEM	Koch-Hill drag model	SKE	E-L approach with deterministic collision mode	Predict the particle motion and pressure gradients in the bed well.
(Almohammed <i>et al.</i> , 2014)	CFD-DEM & Two-Fluid model (E-E)	Hill drag model	SKE	Effect of gas mass flow rate on ability of two simulation techniques to predict flow behaviour.	CFD-DEM model is more successful in predicting realistic flow patterns.
(Adamczyk <i>et al.</i> , 2014)	DDPM (E-L) & Standard	Gidaspow model	SKE	Comparison of standard Euler-Euler and DDPM to model particle transport	- Model modelled comparable results. - Pressure drop exhibit similar tendencies.

Reference	Model	Drag model	Turbulence model	Other specifications	Main Findings
	Euler-euler (E-E)				- Large deviation between experimental and numerical results for the E-E in the bottom of the riser. - E-L proven sufficient to predict modelling particle transport.
(Varas <i>et al.</i> , 2017)	CFD-DEM	Beetstra drag model	-	- The gas phase turbulence was assumed to be insignificant. - Geldart D Particles	- Key clustering characteristics of CFB could be modelled, but number of large clusters over-modelled. - Suggest more research on influence of collision parameters and drag model on clustering.
(Zhu <i>et al.</i> , 2020)	CPFD	EMMS & Combined Wen-Yu & Ergun model	LES	Two drag correlations in predicting flow behaviour under elevated pressure conditions.	- Gas-solids flow shows non-uniform distribution, axially and radially. - EMMS agree well with experimental results. - Wen-Yu/Ergun over-estimate particle dispersion and circulation flux. - Elevated operating pressure enhance gas-solid mixing.
(Zhang & You, 2016)	MP-PIC	Developed PDC model (heterogeneous)	SKE	Heterogeneous drag model developed based on particle distribution characterisation (PDS).	- PDC sufficiently predicts heterogeneous gas-solids flow structures. - PDC prediction on time-averaged distributions and solid fluxes correspond well with experimental results.
(Zhang <i>et al.</i> , 2018)	E-E	EMMS drag model	Not specified	Particles: Geldart B Temperature range: Pressure range:	- Below 0.4 Mpa, changes in solid flux resulted in changes in fluidisation regime. - Increase in pressure reduces solids holdup in the riser.

Reference	Model	Drag model	Turbulence model	Other specifications	Main Findings
(Xie <i>et al.</i> , 2018)	MP-PIC	10 Different drag models investigated	SKE	• Applicability of heterogeneous and homogeneous drag models for CFB modelling. • 1: Geldart A particles Riser diameter: 0.07 m • 2: Geldart B particles Riser diameter: 0.4 m	• Increase in temperature has negligible influence on axial solids holdup, slightly increases solids velocity in core of riser. • Homogeneous model overpredict the solids fluxes and void fraction predictions inconsistent. (1) • Heterogeneous model improved accuracy and could predict core annulus flow. (1) • Both homogeneous and heterogeneous modelled reasonable results. (2)

2.3.4 Closure laws

It is easily seen from the summary of recent CFB modelling studies in Table 2-4, that there exists a wide range of possibilities with regards to models to execute the modelling aspect. The closure of the Reynolds Average Navier Stokes (RANS) equations are crucial for the model accuracy and applicability. The drag model and turbulence model is some of the most important existing closure models.

2.3.4.1 Drag closure model

From the principle of fluidisation, it is easily realised that gravity, pressure gradient and inter-phase drag should be included in the modelling approach to fully capture the flow behaviour in CFBs. The particle phase is coupled to the gas phase by means of an inter-phase drag force to represent the interaction. The drag force can be explained as a large force in the vertical direction to counterbalance the gravitational force on the particle as well as the pressure gradient of the gas. Modelling approaches account for the inter-phase drag force by determining an inter-phase drag coefficient according to different correlations. Furthermore, the accuracy in predicting the drag force between the gas and solid particles has been proven critical for modelling the hydrodynamics in CFBs. Generally, the choice of drag model to predict the drag force and drag coefficient is dependent on the particle Reynolds number and the average void fraction.

Many empirical constitutive models exist to describe the momentum exchange in a homogeneous suspension, but the difficulty arises when applying these models to inhomogeneous suspensions of gas-solid flows. Many researchers dedicate their studies to investigating the effect of different drag models on the prediction of hydrodynamics (Almutterahar & Taghipour, 2008; Benzarti *et al.*, 2012; Ramesh & Raajenthiren, 2010).

The existing drag models can be divided into homogeneous drag models and heterogeneous drag models. Homogeneous drag models require less computational cost, but do not account for the clustering during computation. However, researchers managed to obtain reasonably accurate simulations for fine grids. A few common homogeneous drag closure laws regularly used include the Wen-Yu model, Gidaspow model (Benzarti *et al.*, 2012; Stroh *et al.*, 2018), Clift model, Syamlal O'Brien model (Benzarti *et al.*, 2012; Hartge *et al.*, 2009), Liu model and the Schiller-Naumann model (Almutterahar & Taghipour, 2008). Each of these models has certain constraints and specifications to describe the inter-phase drag force.

- The Schiller-Naumann model is often chosen based on the assumption that all particles are spherical and its ability of predicting a reasonable drag coefficient for a wide range of particle Reynolds numbers (Liu *et al.*, 1993; Loth, 2008; Zhang *et al.*, 2020).
- The Clift model accounts for non-spherical particles, while the Lui model is applicable in the case of deforming particles (Dwivedi *et al.*, 2021; Wu, Chen, *et al.*, 2020).
- The Wen-Yu model is proven to have an optimal performance in dilute particle flows.
- Furthermore, two of the most popular drag models used in CFB modelling, is the O-S drag model and the Gidaspow drag model. Both of these account for the local void fractions in the CFB riser, which is necessary as there exists dense and dilute particle regions in a CFB.

Heterogeneous drag models include the Energy minimisation multiscale (EMMS) model, also the most used heterogenous drag model found in literature (Jiang *et al.*, 2020; Li *et al.*, 2012; Wu *et al.*, 2018; Zhang *et al.*, 2018). The EMMS drag model accounts for the influence of clusters on the drag force exerted on the particles (Gao *et al.*, 2018; Jiang *et al.*, 2020). These considerations led to a significantly improved prediction capability in CFD modelling for CFBs. However, the EMMS drag model is often coupled with an Euler-Euler modelling approach because when used in conjunction with the Eulerian-Lagrangian approach, these models are computationally expensive (Jiang *et al.*, 2020).

In a study by Stroh *et al.*, 2018, they investigated the accuracy of homogeneous drag models and the EMMS drag model using a coarse mesh CFD-DEM approach and further including a particle size distribution (PSD). It was again proven that the EMMS drag model provided a higher degree of accuracy compared to the Gidaspow model, however, the computational effort to produce the similar results was much higher when the EMMS model was employed. A study by Xie *et al.*, 2018, as summarised in Table 2-4, is one of the numerous studies found in literature that attempts to investigate the influence of different drag models on the hydrodynamics. It was found that in most cases the heterogeneous drag models do perform better when compared to the homogeneous drag models, however, they found reasonable agreement with experimental data employing homogeneous drag models.

2.3.4.2 Turbulence modelling of CFB's

Turbulence is responsible for contributing towards a chaotic and irregular flow field and typically exists for higher fluid velocity operations. Turbulent flows are also known for eddies with irregular sizes, resulting in higher momentum, heat and mass transfer rates (Alobaid *et al.*, 2022). Therefore, including and selecting an appropriate turbulence model in any CFD modelling approach is important as it further improves the accuracy of the modelled flow hydrodynamics. In the literature three numerical methods exist to predict and solve the effect of turbulence in

multiphase flow applications (CD-Adapco, 2012). The different models can be categorized into three main approaches:

1. Turbulence models for Reynolds-Average Navier-stokes (RANS) equations
2. Direct Numerical Simulation (DNS)
3. Large Eddy Simulation (LES)

Turbulence models for RANS equations are the most commonly used approach to account for turbulence in a system. This approach includes the k - ϵ model which is widely used by researchers as a result of the simplicity and level of accuracy thereof for a wide range of turbulent flows. A few variations of the k - ϵ model exist, with the mathematical equations adapted for specific applications. The standard k - ϵ (SKE) model solves the average Navier-Stokes equation by solving the turbulent kinetic energy (k) and the energy dissipation rate (ϵ). Further, the turbulent kinetic energy and dissipation rate are used to solve the turbulent eddy viscosity.

Although, turbulence models for RANS equations are extensively used in literature, hydrodynamic studies employing the large eddy simulation (LES) approach are frequently found. The LES approach is based on solving the turbulence scales above a certain threshold directly using a substantially fine grid. The remaining scales below the threshold are solved employing the Smagorinsky turbulence model. The fact that a substantially fine grid is required in the LES approach, it has been proven from various sources that the LES approach requires extensively large computational resources and is therefore not preferred for large scale applications. The same applies for DNS which requires a very fine mesh and adequate time step, leading to an extremely high computational effort to solve all of the turbulence structures.

Even though the LES and DNS methods can produce much more accurate representations, the RANS turbulence models have been more computationally attractive compared to the others, while still producing reasonable results and agreement with experimental data.

2.4 Conclusion

It is clear from the literature survey that studies pertaining to hydrodynamics, CFBs and CFD models, are often validated using experimental work with similar conditions as the CFD models. After validation of the models, the CFD models are then used to describe the hydrodynamics inside a CFB. The models are also often rather validated with results available from literature if no experimental work has been conducted. It became clear that multi-phase flow in CFBs have been successfully described during recent years, especially qualitatively, using the different CFD methods. The topic remains popular amongst researchers continuously focusing on improving the numerical tools available, as well as finding optimal conditions for their specific studies.

However, the quantitative description of multi-phase flow in CFBs still has a lot of room for improvement.

In this study, an attempt is made to develop a CFD model for describing two different sorbent hydrodynamic characteristics in a CFB riser, and consequently validating the developed model with experimental results.

CHAPTER 3 EXPERIMENTAL METHOD

In this chapter, the materials and equipment used to complete the sample preparation, characterisation, and experimental work are introduced and explained. The experimental procedure is described in detail including the start-up and operation of the pilot-scale circulating fluidised bed plant, as well as the method used to take measurements and collect the necessary data.

3.1 Materials and sample properties determination

Experiments are carried out using two different types of adsorbents for the respective SO₂ and CO₂ removal processes. The adsorbents are hydrated lime (Ca(OH)₂) and potassium carbonate (K₂CO₃). Sorbent properties such as particle density and particles size distributions (PSD) are key characteristics needed for calculations and as input parameters in the model development procedure.

To obtain a representative sample, the cone and quartering method was used to reduce the sample size of the respective adsorbents to a manageable size. Thereafter, a rotary splitter was used to further reduce the sample size to sub-samples that are small enough in size to be used in the Malvern Mastersizer 3000. Ethanol was used as the dispersant for PSD analysis and the Helium Pycnometer for particle density. The standards that were used to determine these characteristics are given in Table 3-1.

Table 3-1: Scientific standards used for particle size distribution (PSD) and density analysis

Characteristic	Equipment	Standard
Particle size distribution (PSD)	Malvern Mastersizer 3000	ISO 13321:2020
Particle density	Helium Pycnometer	AS® D854

3.2 Experimental set-up

3.2.1 P&ID and riser dimensions

A laboratory-scale circulating fluidised bed (CFB) facility was constructed for the purpose of FGD research. In this study it was used to conduct the experiments necessary to obtain data for the validation of the generation CFD models. The hydrodynamics in the CFB riser and the influence of various independent variables on the hydrodynamics was investigated using the experimental

plan that is outlined later in this section. In Figure 3-1 below the piping and instrumentation diagram (P&ID) of the facility is given.

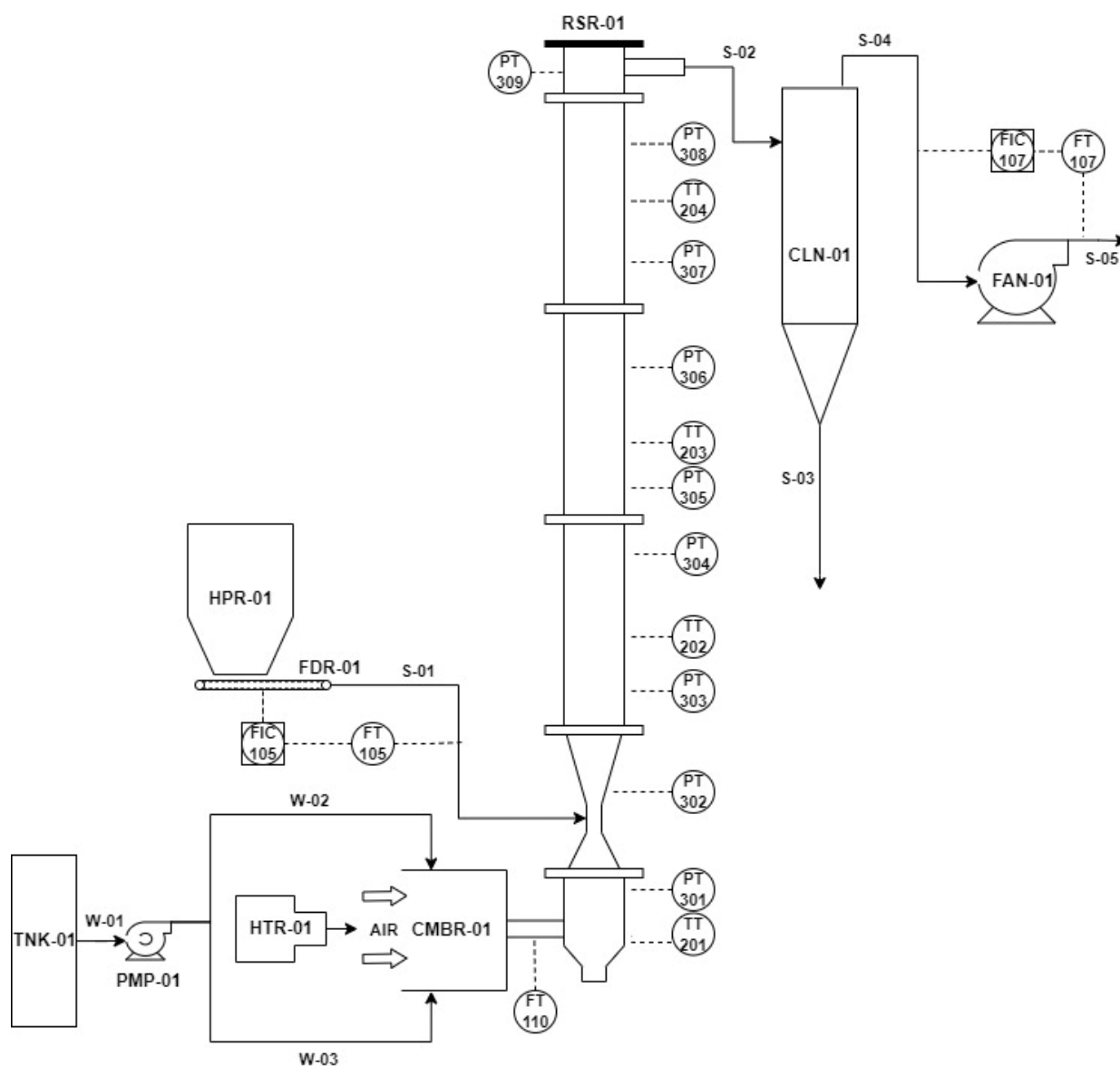


Figure 3-1: P&ID of the experimental facility

The respective adsorbents are loaded into the feed hopper (HPR-01) and fed to the system via a screw feeder (FDR-01). The screw feeder was calibrated prior to the experiments for both the adsorbents. Air, which is used as carrier gas, enters the riser at the bottom and the velocity thereof is manipulated by changing the fan (FAN-01) speed. The fan was pre-calibrated to determine the velocity of the air at a certain operating capacity given as a percentage of the maximum output (%). The calibrated data and curves for both the fan and feeder is available in Annexure A.1. and Annexure A.2.

At the bottom of the riser a chamber (CMBR-01) is constructed to allow for both the heating of the entering air stream and the addition of water to reach and maintain a desired air temperature and

relative humidity level. The air is heated with an oil burner (HTR-01) and the placement of the burner is relative to the entrance of the heating chamber is manipulated to adjust the temperature. Nozzles producing a fine mist-like water feed are used to add the water content to the air stream. Multiple nozzles are used to be able to reach the amount of water needed in the experiments. Calculations have been performed prior to the experiments to determine the amount of water needed to reach the experimental conditions in terms of relative humidity and gas temperature. The calculations were based on the respective days' ambient conditions (dry bulb temperature and relative humidity). The procedure followed are described in Annexure A.3.

At the top exit of the riser, a substantial fraction of the adsorbent particles present is separated from the air by means of a cyclone separator (CLN-01) and collected for re-use. As seen on the P&ID, multiple flush pressure transmitters and resistance temperature detectors were installed to be able to take on-line measurements while experiments take place. The main equipment and measuring instrumentation specifications can be found in Annexure B.1.

Figure 3-2 illustrates the main riser dimensions in a not-to-scale simplified representation.

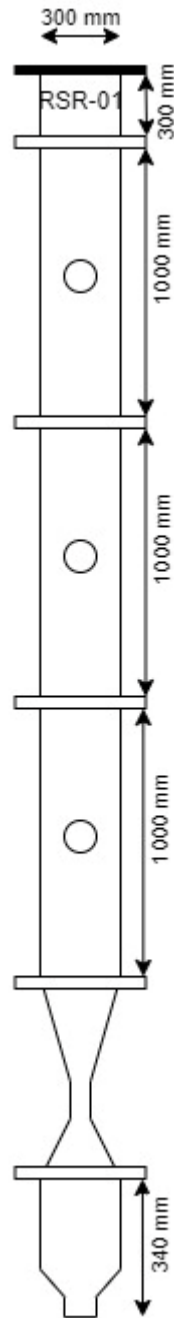


Figure 3-2: Illustration to indicate main riser dimensions.

In Figure 3-3, a schematic illustration is given to indicate the different inlets and outlet, as well as the sampling port and arrangement thereof around the CFB riser. This illustration is used in Chapter 5, to facilitate the discussion of results.

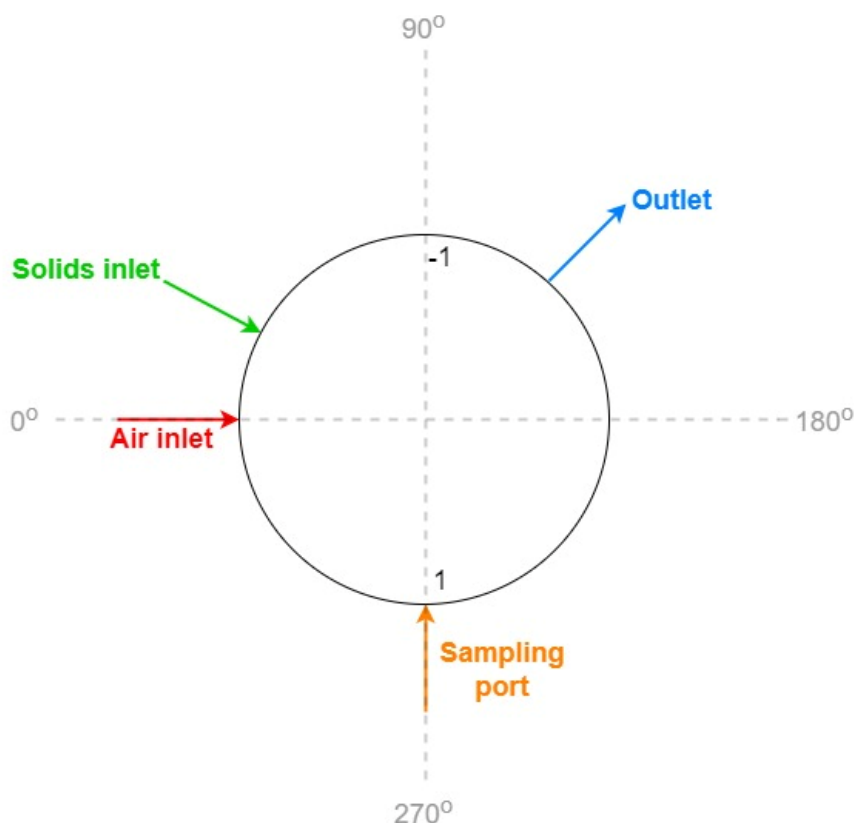


Figure 3-3: Schematic illustration of the different inlets, outlet, and sampling port arrangement

3.3 Experimental plan

In support of the stated objectives, inlet gas temperature and inlet gas relative humidity (RH) are the two key variables that constituted the operating conditions for the investigation. Three different inlet gas temperatures and three different inlet gas relative humidity levels have been chosen in accordance with values found in literature. The respective values are given in Table 3-2 below.

Table 3-2: Experimental variable conditions

Variable	Value
Inlet gas temperature	60°C, 70°C, 80°C
Inlet gas relative humidity	10%, 20%, 30%

Furthermore, variables such as the inlet gas velocity and solids feed rate will be kept constant during all experimental runs. The inlet gas velocity was chosen as 6.5 m/s and the solids feed rate 0.05 kg/s. These values were also used in a previous study where the inlet gas velocity and solids feed rate have been the investigated parameters. The facility was operated at nine different combinations of the two key variables with three repetitive runs of one of the combinations (60°C,

10% RH) for repeatability calculations. This was done for both hydrated lime (Ca(OH)_2) and potassium carbonate (K_2CO_3). The repeatability of the experiments was determined by calculation of the 95% confidence interval. These calculations are presented in Annexure C.

During each experimental run, HPR-01 was loaded continuously to prevent it from running empty, thus influencing the solids feed rate. The fan and screw feeder were switched on and set to the settings that would deliver the air inlet velocity and solids feed rate, which were determined during the pre-calibrations. The heater is switched on and placed before the chamber and lastly, the water pump is switched on. When all equipment was running, a few manual measurements, including the inlet gas temperature and relative humidity were done with the Testo 440 hot wire anemometer to ensure that the experimental conditions are reached. The measurements to be done for data collection were only done after steady experimental conditions have been reached. Steady state was determined after the inlet gas velocity remained within 6.5m/s for 60 seconds.

3.4 Data collection

3.4.1 Calibration of gas inlet velocity

During the initial steps of the experimental program, it was necessary to calibrate the fan as a measure to easily control the velocity during experimental runs. The fan was calibrated by measuring the inlet air velocity at the riser inlet using a Testo 440 hot-wire anemometer. The velocity was measured right at the inlet of the riser, with the tip of the anemometer placed in the middle of the air stream during operation. During operation, the inlet air velocity was displayed on a handheld device, and readings could be recorded and stored, which was used in the calibration calculations. Furthermore, a correction factor was used to adjust the inlet air velocity to account for flow variations across the riser inlet. The calibration procedure and data of the fan are reported in Annexure A.1.

3.4.2 Calibration of sorbent mass flow rate

To determine the sorbent mass flow rate for both hydrated lime and potassium carbonate, the calibration of the sorbent feeder was required. The feeder was calibrated weighing the sorbent discharge for a set time, while the feeder was set to a specific feeder setting (%). The weight was recorded and the sorbent mass flow rate for each setting could be calculated. The calibration procedure, data and curve is reported in Annexure A.2.

3.4.3 Temperature and relative humidity calculations

The method of humidifying the gas (air) stream and controlling the gas temperature required calculations and equipment calibration to determine the water flow rate through the nozzle. The

water spray nozzles (2) were located in the chamber (CMBR-01) downstream of the air-preheater, and upstream of the riser inlet. The required water flow rate through the nozzles was calculated based on air and water properties for the respective temperatures. It was also required to calculate the energy required to heat the air stream from the initial temperature to the final temperature, while evaporation of all the water droplet into the air stream takes place. The calculated energy output required were used in the procurement of an oil burner for this CFB facility. Furthermore, the calculations had to consider the ambient conditions of each day, since the facility was exposed to environmental conditions. In Annexure A-3, the calculations are discussed. Furthermore, manual measurements have been taken throughout execution of the experiments to confirm that the calculations done was correct, and to ensure that the inlet gas temperature and relative humidity are within range from experimental conditions. These measurements were done in the absence of solids particles in the riser due to measuring equipment limitations. The measurements can be seen in Annexure A-4.

3.4.4 Experimental data

Apart from the variables that were adapted for each run to set the operating condition under investigation, the following measurements were conducted inside the riser:

- Pressure
- Temperature
- Solids distribution

Pressure and temperature measurements are recorded using installed flush pressure transmitters and resistance temperature detectors that are indicated on the P&ID (Figure 3-1). The instruments are connected to a PLC and the readings can be stored internally and easily retrieved from the control panel. These measurements are further used to do pressure differential calculations across the entire length of the riser as well as the upper region of the riser as indicated on Figure 3-1. The operating ranges and details of the instrumentation are provided in Annexure B.1.

The riser (RSR-01) consists of three ports across the height thereof. These ports provide access to the inside of the riser and measurements of solids distribution across the diameter of the riser at three different axial positions could be undertaken. A non-isokinetic sampling device is used to collect solid samples at three positions across the diameter of the riser over a time range for 120 seconds. The radial sampling points are illustrated on Figure 3-4 below.

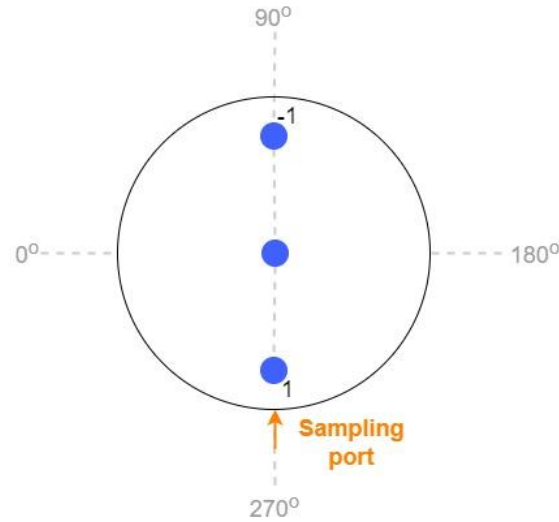


Figure 3-4: Location of radial sampling points

The sampler device is connected to a small, portable vacuum pump, with a gate valve to partially restrict the flow. A toggle valve directly connected to the sampler was used to stop the suction completely while repositioning the sampler or when the sampler Perspex collection bucket was emptied. The samples were weighed, and the masses obtained were further used in calculations. A schematic illustration of the non-isokinetic sampler is given in Figure 3-5.

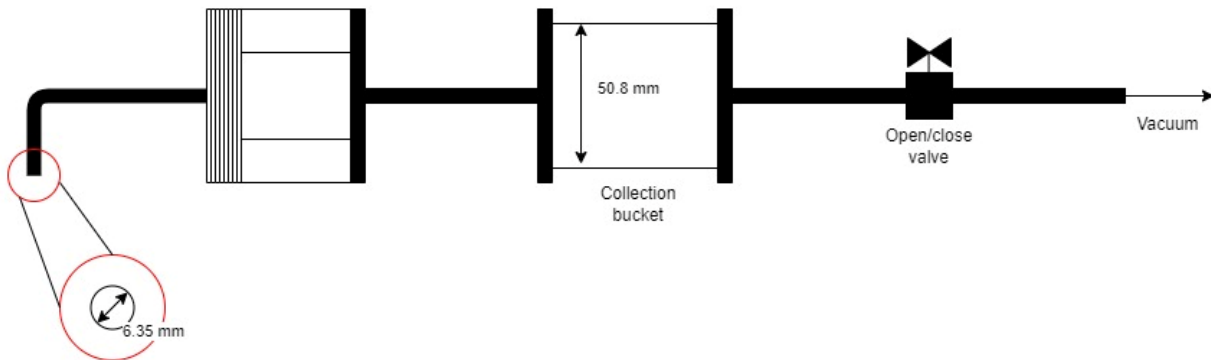


Figure 3-5: Schematic illustration of the non-isokinetic sampling probe

To minimise the effect of the intrusiveness of the sampler, the suction velocity at the tip of the sampler had to be as similar as possible to the gas velocity inside the riser. The sampler suction velocity can be calculated using Equation 3-1. The variation of the sampler suction velocity during operation is negligible and therefore it is assumed that the suction velocity will remain constant during the measurements.

$$u_{\text{sampler}} = \frac{u_{\text{inlet}} A_{\text{inlet}}}{A_{\text{riser}}} \quad (3-1)$$

The mass flow rate of the solid particles in the riser is calculated using the weight of the samples obtained for each measurements together with the time span of 120 seconds, as illustrated in Equation 3-2.

$$\dot{m}_p = \frac{m}{t} \quad (3-2)$$

To summarise the measurements to be taken during experimental runs, Figure 3-6 illustrates a schematic representation of the riser with indications of different measuring points for temperature, pressure, and solids fractions. Figure 3-6, along with Figure 3-4 can be used closely in conjunction with Chapter 5, to facilitate the discussion of results.

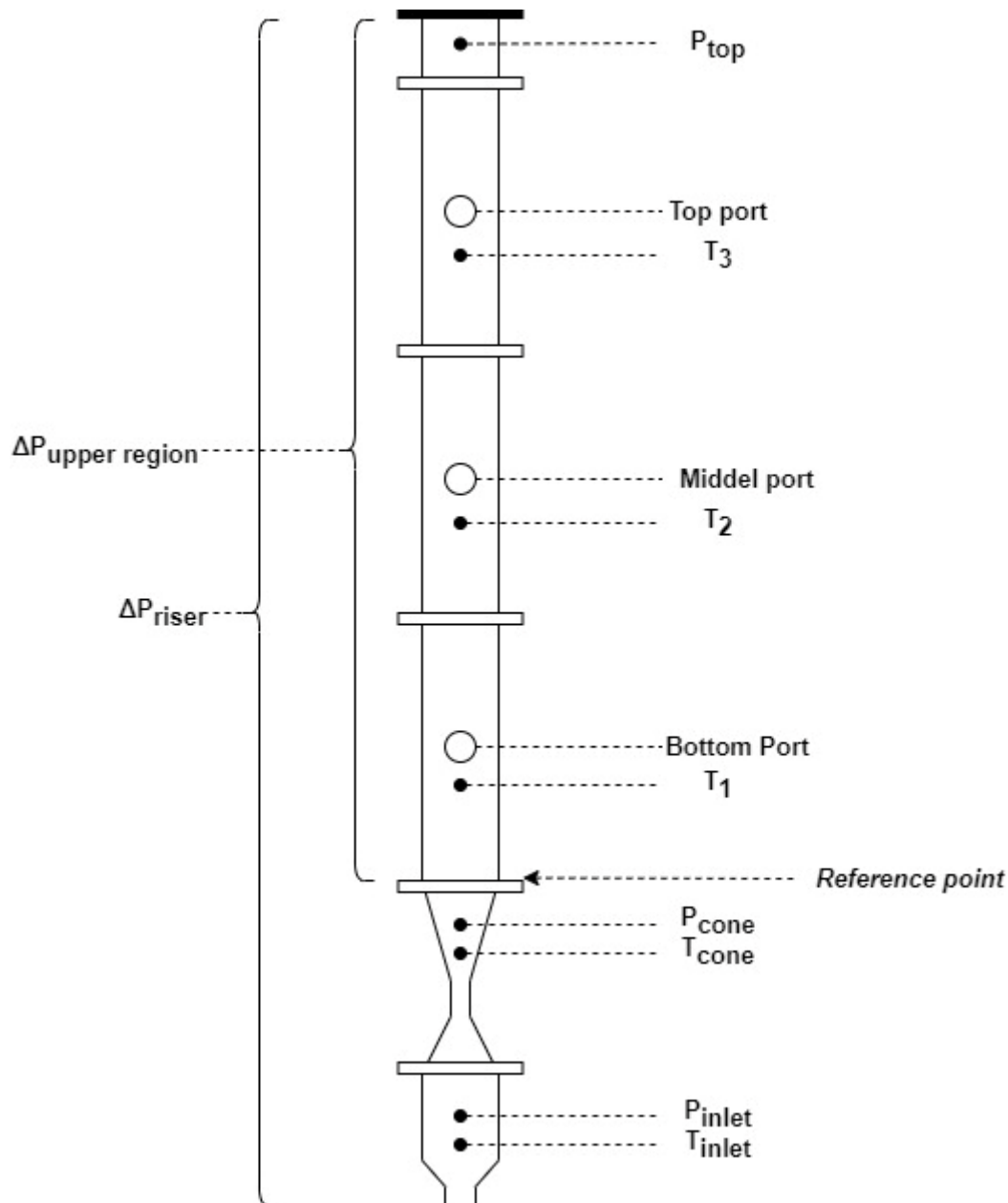


Figure 3-6: Measurement points for temperature, pressure and solids volume fraction across the riser

CHAPTER 4 CFD MODEL DEVELOPMENT

In Chapter 4, the strategy for developing a CFD numerical model for the hydrodynamics analysis of sorbents in a semi-dry CFB riser is described. The model used will be verified with literature information, where after the discretisation and simulation procedure will be discussed. The model has been developed using Siemens STAR-CCM+ software due to its ability to simulate two-phase flow and ability to presents a wide variety of results. The chapter is concluded with a mesh independence analysis.

4.1 Model development

4.1.1 CFD model selection and considerations

4.1.1.1 Governing equations

In the computational fluid dynamic (CFD) environment, the Navier-Stokes equations are widely applied to simulate the flow of incompressible fluids. A fluid is considered to be incompressible for the condition that it has a Mach number smaller than 2, and a very low solid volume fraction. The calculation of the Mach number is based on the fluid flow velocity and is calculated with Equation 4-1 (Welty *et al.*, 2015).

$$M = \frac{u_g}{e} < 0.2 \quad (4-1)$$

Because of the low solids volume fraction and the low gas velocity used in this study, incompressible flow is therefore assumed and the Navier-stokes equations as provided in Equation 4-2 could be used to sufficiently describe the flow (Welty *et al.*, 2015)

$$\rho \frac{d\bar{u}}{dt} = \rho \bar{g} - \nabla P + \mu \nabla^2 \bar{u} \quad (4-2)$$

Here $\bar{u}$ represents fluid velocity, while ρ , μ and P represents the fluid density, dynamic fluid viscosity and pressure respectively. ∇ represents the divergence in the Cartesian coordination system. It is however known that the fluid and flow variables, continuously fluctuate as a result of turbulence (Davidson, 2023; Welty *et al.*, 2015). Computational limitations poses a challenge, as it reduces the mesh capabilities regarding the capturing of process fluctuations. Therefore, it was decided to employ the Reynolds Average Navier Stokes (RANS) equations in this study.

With the assumption of steady, incompressible flow, the RANS equations are expressed with Equation 4-3 with velocity as a variable.

$$\rho \frac{\partial \bar{u}_t \bar{u}_j}{\partial x_i} = -\frac{\partial \bar{P}}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu \frac{\partial \bar{u}_i}{\partial x_j} - \rho \bar{u}'_t \bar{u}'_j \right) + F_B \quad (4-3)$$

In Equation 4-3, μ represents the dynamic fluid viscosity and F_B the body forces exerted per unit volume. Since two-phase flow in CFB's are known to have unsteady behaviour the RANS equation as presented in Equation 4-3, can be modified to the unsteady behaviour, and can be expressed as the unsteady RANS (URANS) equations. The conservation of momentum, energy and mass are then expressed as (Chen *et al.*, 2017; Marocco & Mora, 2013).

$$\begin{aligned} \frac{\partial \alpha_g \bar{u}_i}{\partial t} + \frac{\partial \alpha_g \bar{u}_t \bar{v}_j}{\partial x_i} \\ = -\frac{\alpha_g \partial \bar{P}}{\rho_g \partial x_i} + \frac{\partial}{\partial x_j} \left(\alpha_g \left(v_f \frac{\partial \bar{u}_t}{\partial x_j} - \bar{u}'_t \bar{u}'_j \right) \right) + \alpha_g F_B \\ + F_M \end{aligned} \quad (4-4)$$

$$\frac{\partial(\rho E)}{\partial t} + \nabla(\rho E \bar{v}_g) = F_B \cdot \bar{u}_g + \nabla(\bar{u}_g \cdot \tau) - \nabla \cdot q + S_E \quad (4-5)$$

$$\frac{\partial(\alpha_g \rho_g)}{\partial t} + \nabla \alpha_g \rho_g \bar{u}_g = 0 \quad (4-6)$$

Here ν is the kinematic viscosity of the fluid, F_M is momentum source term, E is the energy per unit mass, q the heat flux and τ the stress tensor respectively. S_E is the energy source term per volume.

4.1.1.2 Turbulence model

To account for turbulence, the three respective approaches as discussed in Chapter 2.3.4.2 exist in which turbulence can be solved. It is also evident from Table 2-4 in Chapter 2.3.3 that the LES and SKE turbulence models are the most widely used turbulence model in literature. From the discussion and literature findings, it was decided to use turbulence models that are developed for the RANS equations such as the standard $k - \varepsilon$ turbulence model.

The influence and contribution of turbulence to the total shear stress tensor of the fluid are represented by the Reynolds stress tensor, $\bar{\tau}_f$, and can be expressed by the Boussinesq hypothesis as represented in Equation 4-7 (Smolders & Baeyens, 2001).

$$\tau_{ij} = -\bar{U}'_t \bar{U}'_j = -\frac{2k}{3} + \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (4-7)$$

Here k represents the turbulent kinetic energy and μ_t the turbulent eddy viscosity. The calculation of turbulent eddy viscosity (μ_t) in Equation 4-7 above results in the closure of RANS equations. To calculate the turbulent eddy viscosity, the influence of eddies present in turbulent flow on the flow profile in a CFB was approximated by implementing the applicable turbulent model as discussed previously in Chapter 2.3.

There exist numerous methods to determine the turbulent eddy viscosity, however, lately the incorporation of transport equations to solve for turbulent eddy viscosity has become more popular as it results in an improved accuracy compared to the initial algebraic expressions used to solve turbulent eddy viscosities (Argyropoulos & Markatos, 2015).

The standard $k - \varepsilon$ (SKE) turbulent model is a two-equation model with a partial differential equation (PDE) for both the turbulent kinetic energy and the energy dissipation rate (Argyropoulos & Markatos, 2015; Liu *et al.*, 2018). The turbulent eddy viscosity can then be expressed as:

$$\frac{\partial}{\partial t}(\rho k) + \nabla(\rho k \bar{u}) = \nabla \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right] + P_k - \rho(\varepsilon_d - \varepsilon_{d0}) + S_k \quad (4-8)$$

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \varepsilon_d) + \nabla(\rho \varepsilon_d \bar{u}) \\ = \nabla \left[\left(\mu + \frac{\mu_t}{\sigma_{\varepsilon_d}} \right) \nabla \varepsilon_d \right] + \frac{\varepsilon_d}{k} C_{\varepsilon_{d1}} P_\varepsilon - C_{\varepsilon_{d2}} f_2 \rho \left(\frac{\varepsilon_d^2}{k} - \frac{\varepsilon_{d0}^2}{k} \right) \\ + S_{\varepsilon_d} \end{aligned} \quad (4-9)$$

$$\mu_t = \rho C_\mu f_\mu \frac{k^2}{\varepsilon_d} \quad (4-10)$$

In these equations k represents turbulent kinetic energy, ε_d is the energy dissipation rate, σ_k , σ_{ε_d} , C_μ , $C_{\varepsilon_{d1}}$, $C_{\varepsilon_{d2}}$ are model constants and P_k and P_ε are production terms. S_k and S_{ε_d} are the source terms.

Lastly, general modification to the SKE-model exist to improve the model capability to predict turbulence in a flow environment where a pressure gradient largely exists, as it was found to perform poorly in such an environment (Argyropoulos & Markatos, 2015). The modified version is known as the realisable- $k - \varepsilon$ turbulence model.

The realisable $k - \varepsilon$ turbulence model has two major differences from the SKE being the equations to solve the energy dissipation rate and the method to calculate the turbulent eddy viscosity. Equations 4-11, 4-12 and 4-13 present the set of equations used to solve the energy dissipation rate (Lateb *et al.*, 2013; Van Maele & Merci, 2006).

$$\begin{aligned}
& \frac{\partial}{\partial t}(\rho \varepsilon_d) + \nabla(\rho \varepsilon_d \bar{U}) \\
&= \nabla \left[\left(\mu + \frac{\mu_t}{\sigma_{\varepsilon_d}} \right) \nabla \varepsilon_d \right] + C_{\varepsilon_{d3}} S_{\varepsilon} \\
&- C_{\varepsilon_{d2}} \frac{k}{k + \sqrt{\bar{U} \varepsilon_d}} \rho \left(\frac{\varepsilon_d^2}{k} - \frac{\varepsilon_{d0}^2}{k} \right) + S_{\varepsilon_d}
\end{aligned} \tag{4-11}$$

$$C_{\varepsilon_{d3}} = \max \left(0.43, \frac{\eta}{\eta + 5} \right) \tag{4-12}$$

$$\eta = \frac{S(k)}{\varepsilon_d} \tag{4-13}$$

The turbulent eddy viscosity in this method is approximated with Equation 4-14 rather than using a constant (Lateb *et al.*, 2013; Van Maele & Merci, 2006).

$$C_{\mu} = \left(A_0 + A_s \bar{U}^* \frac{k}{\varepsilon_d} \right)^{-1} \tag{4-14}$$

Here $A_0 = 4.04$ with A_s , σ , W , S , $\bar{U}^*$ and $\bar{\Omega}_{if}$ calculated with different expressions. The above modifications to the SKE provide improved predictions of turbulence in flows, but as a result of the flow separation being insignificant, the SKE equations were proven to be sufficient for predicting the turbulence in this study.

4.1.1.3 Drag model

The drag model applicable to the simulations would largely depend on the coupling method used. In the consideration of one-way and two-way coupling the inter particle would be negligible when compared to the gas-particle interactions as discussed previously in Chapter 2.3. As a result, the particle volume fraction's influence on the drag force is usually neglected. Furthermore, the often-used assumption that all particles are spherical in shape to simplify the model and reduce computational effort, makes the Schiller-Naumann drag model a sufficient model to use. Therefore, from a group of homogeneous drag models, the Schiller-Naumann drag model was chosen for this study (Alobaid *et al.*, 2022). This model is capable to provide a drag coefficient within 5% from the standard drag coefficient for particles with a Reynolds number between 800 and 1000 (Alobaid *et al.*, 2022; Rhodes, 2008). The Schiller-Naumann drag coefficient can be expressed by Equation 4-15.

$$C_D = \begin{cases} \frac{(1 + 0.15R_{fp}^{0.687})}{R_{fp}} & \text{for } R_{fp} \leq 1000 \\ 0.44 & \text{for } R_{fp} \geq 1000 \end{cases} \quad (4-15)$$

The Schiller-Naumann drag model can be categorised as a homogenous drag model. These types of models do not consider the clustering of particles. However, fairly accurate results can be achieved when using an extremely fine mesh. Some other popular homogenous drag models that exist are the Koch-Hill, Liu, Wen-Yu, Gidaspow and Syamlal & O'Brien (O-S) drag correlations as discussed previously in Chapter 2.3. The Gidaspow and O-S models are the most suitable for gas-solid flows in CFB as it accounts for local void fractions. Furthermore, it is the Gidaspow model that would be the preferred model to employ in particulate flows in a CFB from findings in literature (Chen *et al.*, 2017; Jiang *et al.*, 2020; Ramesh & Raajenthiren, 2010; Wu *et al.*, 2018; Xie *et al.*, 2018).

4.1.1.4 Heat transfer model

Heat transfer models are often coupled with simulation approaches to further improve the accuracy of the modelled hydrodynamics in high temperature applications. The general assumption exist that a single particle transmits heat fluxes through radiation, convection and conduction. Convection occurs between the particle and fluid, and conduction between particles. Since the temperature range used in this study is not exceptionally high, radiation can be ignored as it is proved to have a negligible contribution in simulation. (Ding *et al.*, 2008; Ellendt *et al.*, 2018; Wang & Zhu, 2016; Welty *et al.*, 2015). The three heat transfer mechanisms; particle-particle conduction, particle-fluid-particle conduction and particle fluid convection has previously been verified for the application in CFB by (Uwitonze *et al.*, 2022; Wang *et al.*, 2010). The equation to express the three mechanism are provided in Equation 4-16, 4-17 and 4-18 below.

$$Q_{fp,i} = h_{pf,i} A_{p,i} (T_f - T_{p,i}) \quad (4-16)$$

$$Q_{pp,ij} = \frac{4k_{p,i}k_{p,j}}{k_{p,i} + k_{p,j}} (T_{p,j} - T_{p,i}) \sqrt{R_{p,j}^2 - \left(\frac{R_{p,j}^2 - R_{p,i}^2 + I_{ij}^2}{2I_{ij}} \right)^2} \quad (4-17)$$

$$Q_{pfp,ij} = k_g (T_{p,j} - T_{p,i}) \int_{R_{in}}^{R_{out}} \frac{2\pi r}{I_{ij} - \left(\sqrt{R_{p,i}^2 - r^2} + \sqrt{R_{p,j}^2 - r^2} \right)} dr \quad (4-18)$$

4.1.2 Geometry discretisation and solution algorithm

The flow of fluids and solids are generally described by partial differential equations (PDEs) and solved using existing numerical methods. The solution approach can be categorised as either the finite difference method (FDM), finite element method (FEM) or the finite volume method (FVM). The FVM is predominantly employed due to its independence from the mesh structure. Solvers in STAR-CCM+® solve the PDEs using the FVM which is based on the integral form of the modelling equations (CD-Adapco, 2012). The FVM performs three primary steps to provide a solution, which are:

1. Mesh generation,
2. Discretisation of PDE's and,
3. Solving the modelling equations.

During mesh generation the geometry is separated into numerous discrete control volumes. These control volumes typically have either hexahedral, tetrahedral, or polyhedral geometry. Figure 4-1 illustrates these three distinct geometries respectively. Each of these geometries have several advantages and disadvantages however, the polyhedral meshes combine the advantages of the hexahedral and tetrahedral meshes. Polyhedral meshes are considered the most suitable to use for more complex geometries. The main advantage of polyhedral meshes is the ability to be stretched to further lowers the number of control volumes required and the numerical diffusion associated herewith (Chen *et al.*, 2021; Sosnowski *et al.*, 2017).

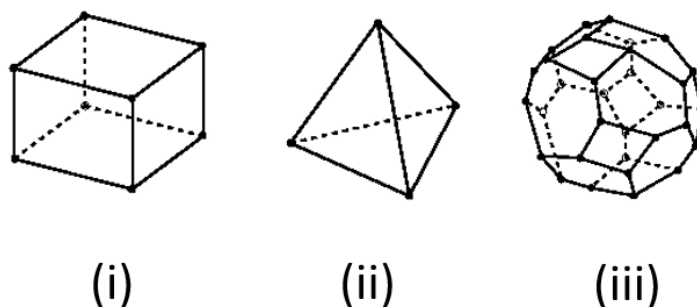


Figure 4-1: Different cell structures used in CFD discretisation: (i) Tetrahedral, (ii) Hexahedral and (iii) Polyhedral mesh structures
adapted from (Sosnowski *et al.*, 2017)

The discretisation of PDE's as discussed in Section 4.1.1 entails the equations to be integrated across each control volume for a discretised numerical solution. The variables' respective values and gradients are generally approximated using the simplest interpolation scheme, which is central differencing. Finally, solving the discretised equation is approached by employing the

semi-implicit method for pressure linked equations (SIMPLE) algorithm. A representation of the SIMPLE algorithm is presented in Figure 4-2.

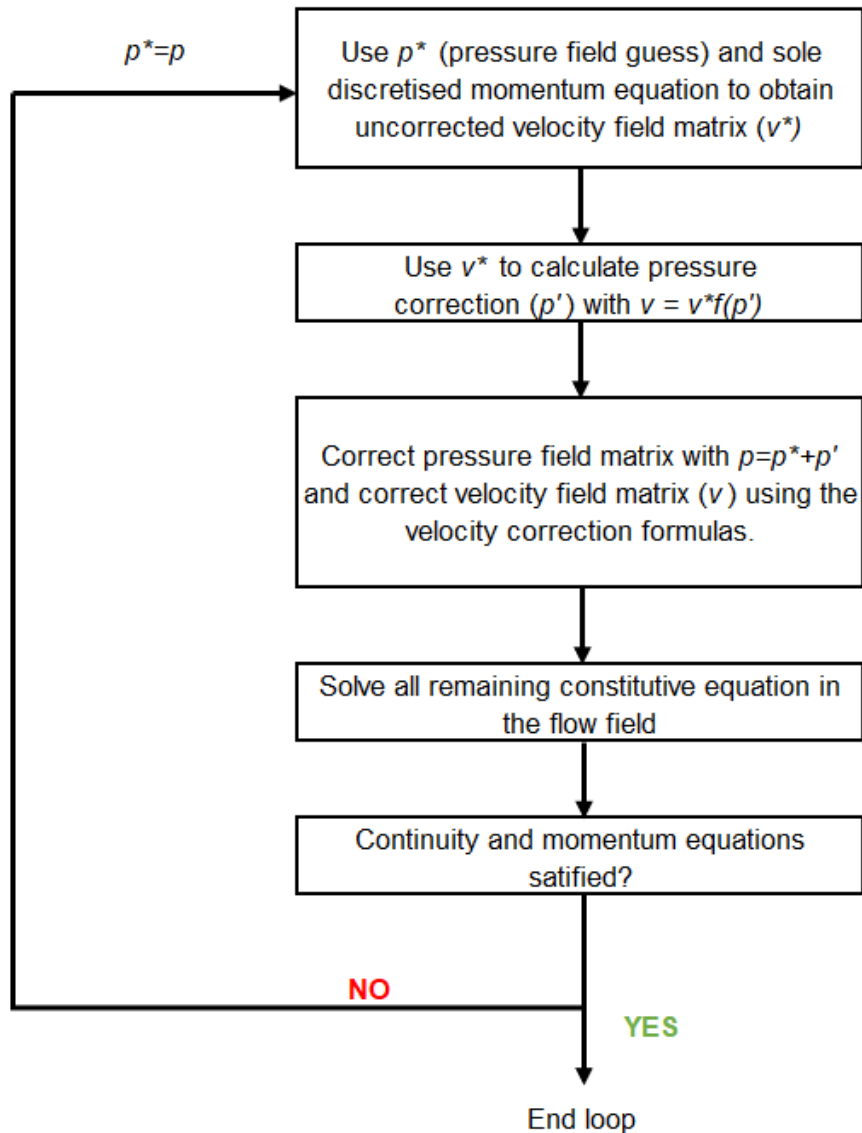


Figure 4-2: Flow diagram illustrating the SIMPLE algorithm
Adapted from (Versteeg & Malalasekera, 2007)

Throughout literature the SIMPLE algorithm is the most widely used segregated solver. It was also chosen for this study. This algorithm initially guesses a pressure field. Subsequently, the pressure field is used to solve the momentum equations in order to yield a velocity field matrix. The velocity field matrix is then used to calculate pressure correction terms. The pressure correction term is used to update the pressure field matrix and the velocity field matrix is used to close all other equations. If the continuity equation is not satisfied, the pressure field matrix

becomes the new “guessed” pressure field and the algorithm is repeated until the continuity equation is satisfied.

The algorithm can further be adjusted for the simulation of transient conditions. An outer loop is added for time marching. A time step is chosen after which the algorithm is allowed to converge. The time is updated after convergence is achieved and a given number of inner iterations is completed whereafter the process is repeated until the defined time constraint is satisfied.

The adjusted SIMPLE algorithm is presented in Figure 4-3. The algorithm steps after the adjustments are as follows:

1. Initialise pressure field and velocity field matrix;
2. Solve Newton’s second law of motion for each parcel in the Lagrangian phase;
3. Run the SIMPLE algorithm;
4. Update time step and;
5. Repeat until the time step is satisfied.

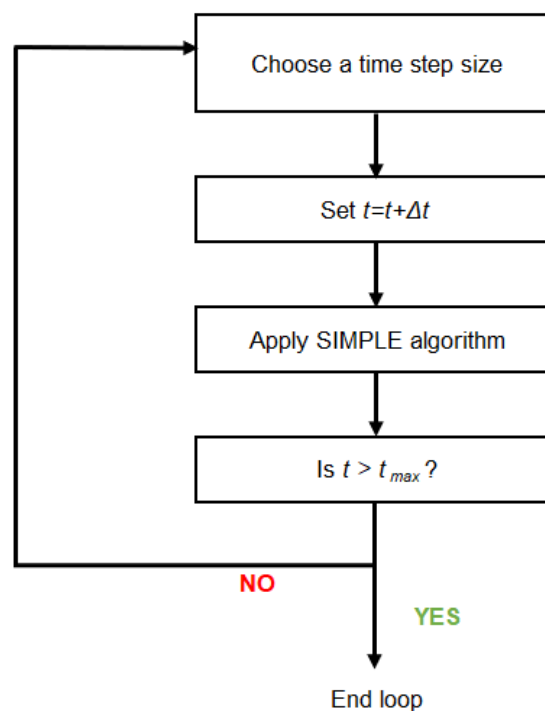


Figure 4-3: Adjusted SIMPLE algorithm
Adapted from (Versteeg & Malalasekera, 2007)

4.2 CFD simulation procedure

4.2.1 Computer-aided design (CAD) of the CFB riser

The initial step in any model development study requires the construction of a 3-D computer-aided design (CAD) model to define the geometry and boundaries of the equipment. Siemens NX-12[®] software was used to construct the geometry of the CFB riser with dimensions equivalent to the experimental facility (Siemens, 2017). The dimensions of the riser are given in Figure 3-2. The region of fluid flow was presented as a single, solids body, as Star-CCM+[®] recognises an imported solid body as a fluid flow region, unless it is specifically specified. The CAD model is prepared and the Parasolid file is subsequently imported into STAR-CCM+[®] for model development to commence. The CFB riser geometry as imported into the modelling software is shown in Figure 4-4.



Figure 4-4: CFB Riser geometry

4.2.2 STAR-CCM+® simulation procedure

Once the CAD model was imported into the STAR-CCM+® software, it automatically creates a part with a single surface area. The surface was then discretised by splitting the riser geometry into its separate part curves namely gas inlet, particle inlet, wall, and outlet. These four distinct parts were each assigned to a specific region and a boundary was created for each of the part surfaces. Both the gas inlet and particle inlet were specified to be stagnation type inlets. Stagnation inlets allow for the inlet gas to have an initial state of rest. The negative gauge pressure at the outlet was varied to achieve the desired inlet gas velocity at the bottom of the riser. To be able to do this variation, the outlet was specified as a pressure type outlet. The wall boundary was specified as a wall with roughness height of 0.02 mm; a value typically used in literature for risers manufactures from similar materials (Welty *et al.*, 2015).

The physics models within the simulation package were chosen in accordance with literature and the model verification in Section 4.1.1. The model considers two-phase flow within a Eulerian-Lagrangian framework, where the fluid region was specified as a gas phase and represents a humid, warm gas. The RANS equation could sufficiently be implemented to describe the gas phase flow as the Mach number is less than 2. The simulation was solved in the three-dimensional (3-D) domain and the non-reacting, segregated flow model was used. Turbulence was accounted for by selecting the $k - \varepsilon$ turbulence model given the fact that the model has a significantly small quantity of flow separation in the CFD added to its ease of computation. Furthermore, the segregated fluid temperature model was selected. The implicit unsteady solver was chosen for this simulation, and the algorithm applied by this solver is discussed in Section 4.1.2.

The initial gas and particle properties, together with constants of interest are stipulated in Table 4-1. Specifically, the wall normal and tangential restitution coefficients were obtained from a literature study for particles ranging from 10 μ m to 500 μ m (Sommerfeld & Huber, 1999).

Table 4-1: Initial gas and particle properties and constants used in STAR-CCM+® simulations

Constant/property	Value
Density of air (@ 22.9°C)	1.19 kg/m ³
Dynamic viscosity of air (@ 22.9°C)	1.83 x 10 ⁻⁵ Pa·s
Temperature of air	60°C/ 70°C/ 80°C
Particle diameter of Hydrated Lime	D[4, 3] = 30 μ m
Particle density of Hydrated lime	2376 kg/m ³

Particle diameter of Potassium Carbonate	$D[4, 3] = 82 \mu\text{m}$
Particle density of Potassium Carbonate	1720 kg/m^3
Particle mass flow rate	0.05 kg/s
Normal wall restitution coefficient	0.97
Tangential wall restitution coefficient	0.94
Gravity	9.81 m/s^2

As discussed in Section 4.1.1, a homogenous drag model was selected for the simulation and in this case the Schiller-Naumann drag model was employed. Since the particle velocities never exceeded nor reached 520 m/s which corresponds to a Reynolds number of 800 for the specific particles, the Schiller-Naumann model is sufficient in predicting the drag coefficient.

The particle inlet was chosen to be the point of entry for the particle and a surface injector was chosen. The direction of entry was normal to the particle inlet. An injection point density of 28,000 was used with a particle size according to the average $D[4,3]$ as determined during analysis. The inlet mass flow rate was specified to resemble the experimental value.

Obtaining valuable results from a simulation is extremely important and there exist numerous ways to present the results required to make comparisons and draw conclusions. In hydrodynamics studies, tracking the trajectories of the particles can provide a lot of valuable insight into particle behaviour. To be able to track these trajectories, the timing parameters need to be specified. The Courant numbers, being a non-dimensional number evaluating the required time step for the chosen mesh size and flow velocity, were specified as 0.05 and 0.35 respectively.

Scenes (scalar and vector visualisations) were created to view the properties such as, particle velocity, gas velocity, pressure and temperature profiles. In each case, the respective part and property has been chosen to create all the different scenes. Different plane sections in the x-, y- and z-axis and line probes are used to collect data on the exact same points as during the experiments. XY-plots have been used to facilitate comparisons between experimental and modelled data.

After all the simulation inputs have been selected and specified, the simulation is ready to be solved. The CFD model is initially started to be solved in steady state conditions. The simulation is allowed to solve for 500 iterations with only the gas-phase present in the riser. The solid particles are only injected after the 500th iteration. This method is used to reduce the overall simulation time by providing better initial values for the Lagrangian phase solver and the implicit unsteady solver. The steady solver was replaced by the implicit unsteady solver after the 500th

iteration. The inner and outer simulation iterations were specified as 5 and 3000 respectively, which is the number of iteration that the SIMPLE algorithm is allowed to converge and the number of times the algorithm is repeated before the time step is updated.

4.3 Mesh independence analysis

A mesh independence analysis validates that the influence of the mesh quality and properties on different process parameters is negligible. It is also performed to minimise discretisation errors and the computational time. This enables a solution to achieve an optimal ratio between solution and simulation time.

The mesh quality in this study is quantified based on three cell metrics that can be extracted. The applicable metrics are (Chen *et al.*, 2020):

1. Cell skewness angle: the angle between the normal cell face and the connection line between the cell centroid with the adjacent cell. When 90° , there exist convergence issues as the dot product within certain denominators becomes 0. Ideally the skewness angle should be below 40° .
2. Volume change ratio: the ratio of the volume of specific a cell to that of its neighbouring cells. It is known to create stability issues within the simulation when the ratio becomes very small. A good volume ratio should be more than 0.5.
3. Cell quality: this cell metric indicates the cell non-orthogonality. A cell quality of approximately 0 would indicate an inadequate cell. The cell quality should ideally be larger than 0.5.

Based on the above explanation of the three cell metrics used in this study, a criteria was created to identify invalid cells. Furthermore, three different mesh base sizes were chosen for the mesh independence ranging from 25 mm to 60 mm. By decreasing the mesh size global mesh refinement could be achieved. Two of the grids were further refined in the bottom of the riser, to capture the large temperature and pressure gradients in the cone area of the riser. The local refinement was achieved by creating volumetric controls within STAR-CCM+[®]. This way the properties of the mesh in the bottom region could be varied from that in the upper region. Table 4-2 summarises the different grids that were used in the study.

Table 4-2: Mesh properties for mesh independence analysis

Grid	Mesh base size (mm)	Local refinement (Yes/No)	Volumetric control base size (mm)
60 mm (G)	60	No	-
40 mm (G)	40	No	-

25 mm (G)	25	No	-
60 mm (L)	60	Yes	15
40 mm (L)	40	Yes	10

*where G = Global refinement and L = Local refinements

In the analysis it was seen that the number of invalid cells was significantly reduced with the addition of local mesh refinement in the case for the 60 mm (L) mesh, while the effect was less prominent for the 40 mm (L) mesh as presented in Figure 4-5. Furthermore, the analysis based on the criteria for the three cell metrics gave the results as presented in Figure 4-6. When comparing the mesh quality according to the criteria given earlier, the 40 mm (L) mesh has the largest percentage cells with good metrics in all three cell metrics. Since the 25 mm (G) mesh has a very low number of cells for the volume change category, this mesh will not be considered further in the analysis, but will be used as reference for the analysis.

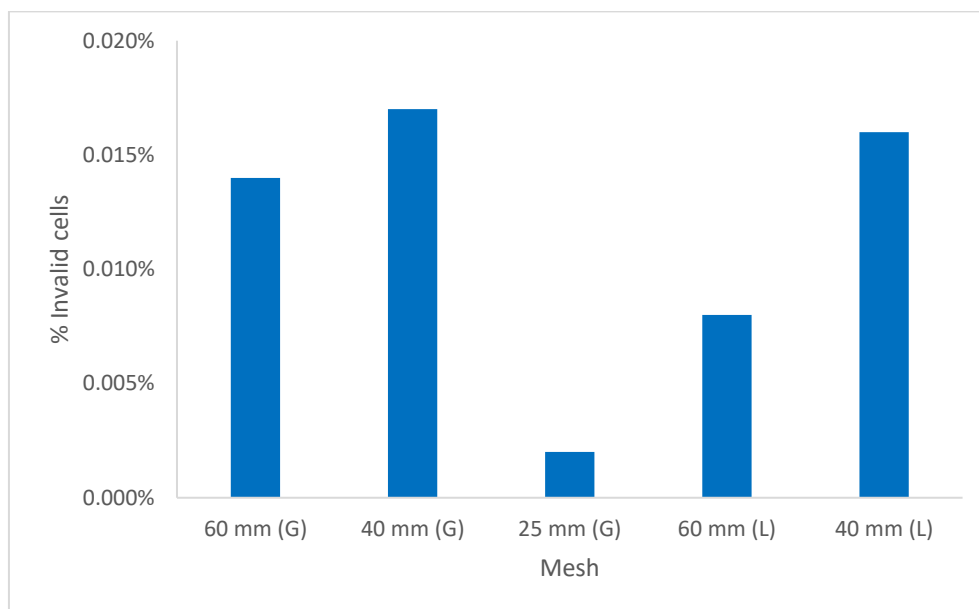


Figure 4-5: Percentage invalid cells present in each mesh

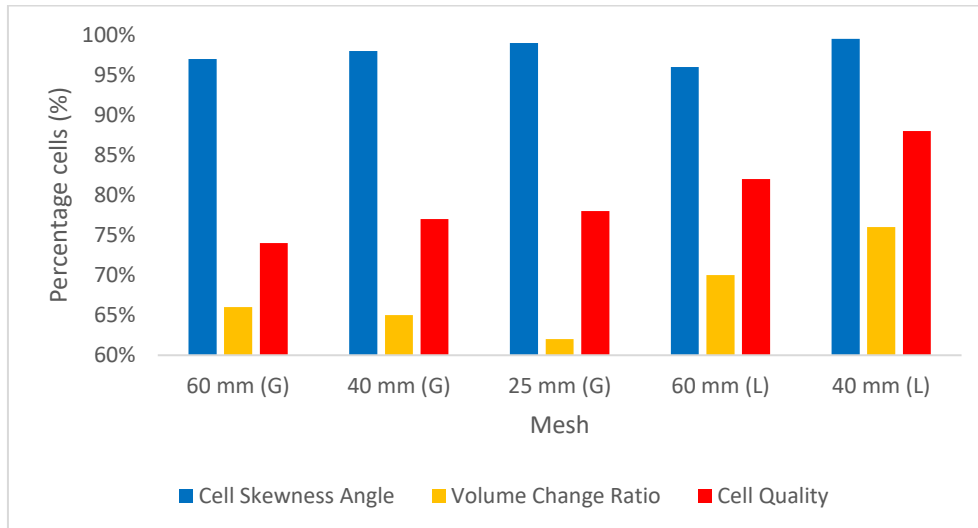


Figure 4-6: Percentage cells with good cell metrics in each category

The performance of each of the four remaining meshes were compared for the solution time, the modelled pressure as well as solids volume fraction at the three ports. The simulation was allowed to run for 5 seconds of riser operation whereafter results were retrieved. The data retrieved was only for reference on the performance of the different meshes and should not at any time be used for comparison with experimental results. Illustrated in Figure 4-7 is the time required to simulate 5 seconds of riser operation for each mesh.

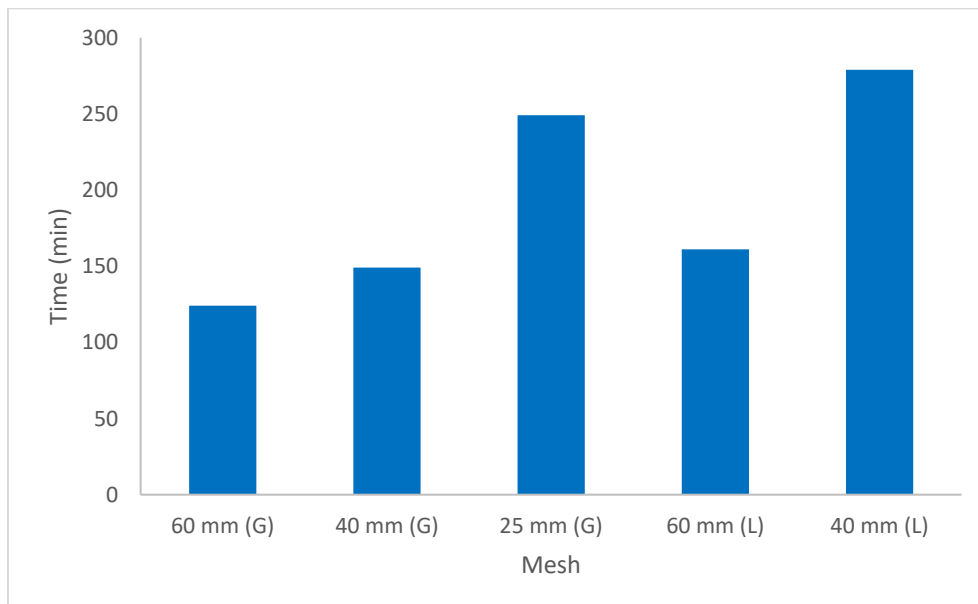


Figure 4-7: Time required to simulate 5 seconds of riser operation for each different mesh

It can clearly be seen that for a global finer mesh the addition of local mesh refinement significantly increases the required simulation time. The time increased from 149 minutes for a 40 mm global mesh to 279 minutes for a 40 mm mesh with local refinement in the bottom of the riser. For the 60 mm mesh the time increased from 124 minutes to 161 minutes.

The modelled pressure at the top of the riser for each mesh is illustrated in Figure 4-8. In both meshes a local refinement resulted in an increase in pressure. The difference was calculated to be approximately 5%.

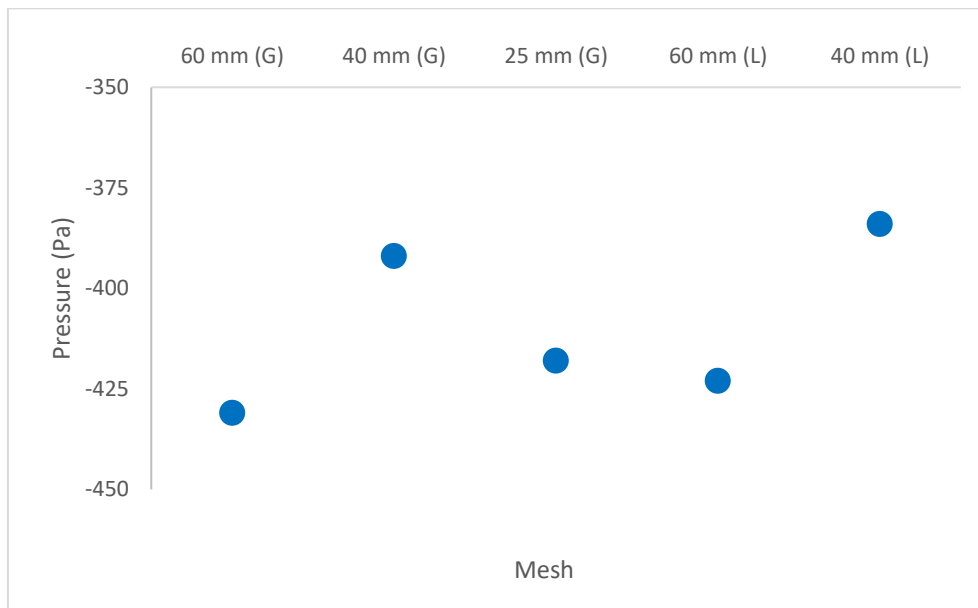


Figure 4-8: Modelled pressure at the top of the riser for each different mesh

Lastly, the average solids volume fraction at the three riser ports for each mesh is depicted in Figure 4-9. It is seen that the global 60 mm mesh exhibits the greatest deviation from the 25 mm mesh. The trend that was modelled by the 25 mm mesh could not be seen in any of the other meshes. On the contrary, three of the four meshes modelled the top port to have the highest average solids volume fraction.

Considering all observations, the 25 mm mesh was eliminated from being used in this study due to it having the lowest percentage cells with good volume change ratio metrics. The 40 mm mesh with local refinement was eliminated from this study due to it having the highest required simulation time. From the remaining meshes the global 40 mm mesh showed the greatest deviation in modelled pressure when compared to the global 25 mm mesh. The average solids volume fraction for the global 60 mm mesh exhibited the opposite trend when compared to the 25 mm global mesh and was eliminated as a result. Therefore, the 60 mm mesh with local refinement mesh of 15 mm was selected for use for all simulations in this study.

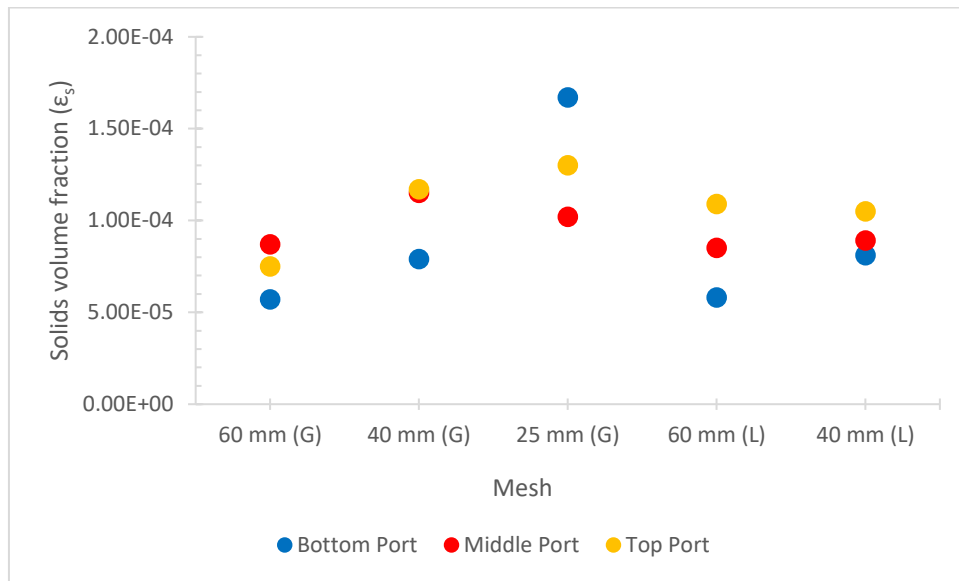


Figure 4-9: Modelled average solids volume fraction at three riser ports for each different mesh

In Figure 5-10, the final mesh structure obtained is illustrated.

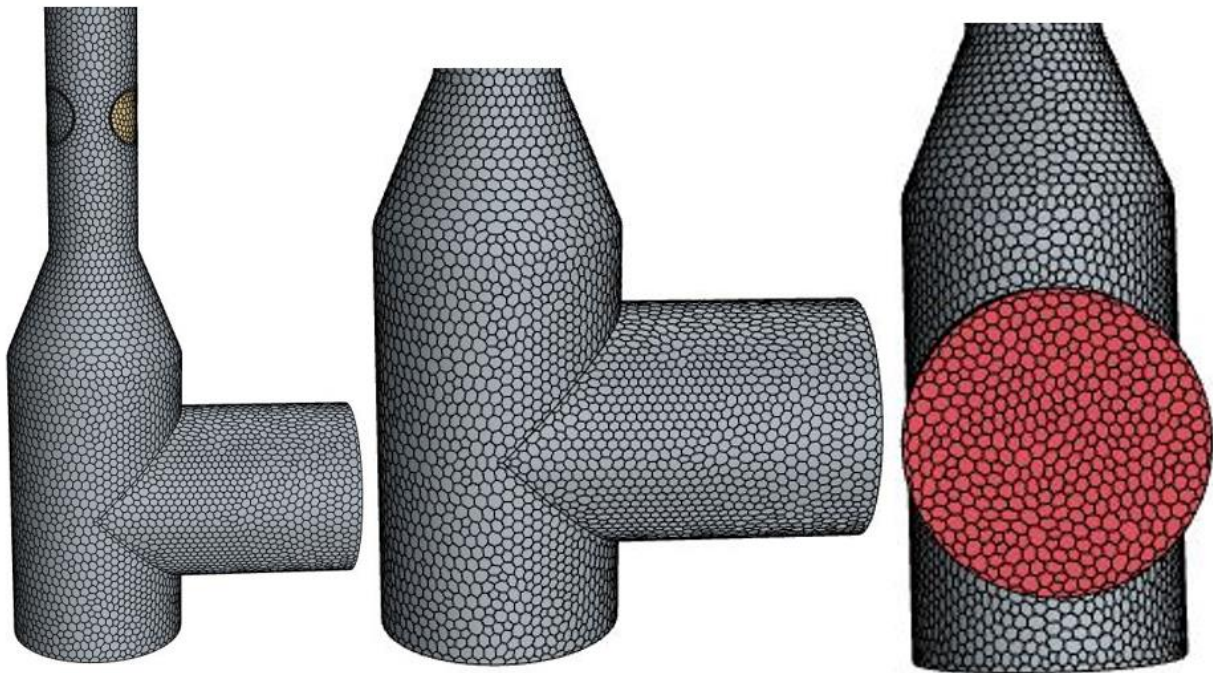


Figure 4-10: Mesh structure in CFD modelling

CHAPTER 5 RESULTS AND DISCUSSION

In Chapter 5, all the data collected during the study are brought into focus and is presented in such a way that conclusions can be made. The chapter starts off with the results from determining certain properties of the sorbents (Section 5.1). It is followed by the validation of the CFD developed model as developed in Chapter 4 (Section 5.2) and finally the results from the experimental runs and simulations are discussed together in Section 5.3. The chapter is concluded with a swift comparison of results between hydrated lime and potassium carbonate as adsorbents for sulphur dioxide and carbon dioxide respectively.

5.1 Solid sorbent properties

The two sorbents investigated, namely hydrated lime and potassium carbonate, are respectively used in SO₂ and CO₂ capturing processes due to their excellent absorbing properties. For this study, the particle size ranges and particle densities were identified to be important input information for modelling of the hydrodynamics and therefore these properties were determined with the previous mentioned analysis techniques as discussed in Chapter 3.1. The results of the two respective sorbents are tabulated in Table 5-1.

Table 5-1: Solid sorbent properties

Sorbent property	Sorbents	
	Hydrated Lime (Ca(OH) ₂)	Potassium carbonate (K ₂ CO ₃)
Particle size (d_p)	D [3, 2] = 7.47 μm	D [3, 2] = 38.2 μm
	D [4, 3] = 30 μm	D [4, 3] = 82 μm
Particle density (ρ_p)	2376 kg.m^{-3}	1720 kg.m^{-3}

In hydrodynamic studies, the solid materials being fluidised are usually classified according to the Geldart classification based on their behaviour during fluidisation. According to the results, hydrated lime would classify as a Geldart Group C powder. These powders typically have a Sauter mean diameter (D [3, 2]) of less than 30 μm and are the most difficult group to fluidise. These particles are cohesive and tend to behave more like particle clusters than single particles.

The potassium carbonate falls within the Geldart Group A powders. These powders are known to be the preferred choice of powder types used in fluidised beds because of their easy fluidisation properties. The Sauter mean diameter of group A classified particles ranges from 30 μm to 125 μm , with particle densities in the order of 1500 kg.m^{-3} . Furthermore, the De Brouckere mean

diameter (D [4, 3]) were used as input in the modelling as it reflects the average particle size which constitutes to the majority particles of a sample.

5.2 Validation of CFD model with experimental data

To evaluate the efficiency of using a CFD model as described in Chapter 4 to predict and study the hydrodynamics of hydrated lime and potassium carbonate in a CFB, it is beneficial to validate simulation results with experimental data. In this study, experimental data was collected from performing experimental runs under the same conditions that are used in the simulations. Given that a substantial amount of data was collected from both experiments and CFD modelling, only a some of data sets were selected for validation purposes.

5.2.1 Hydrated lime model validation

In Figure 5-1, the experimental and modelled pressure differential across the riser is given. The percentage deviation of the modelled pressure differential from the experimental values has been calculated with Equation 5-1 and is illustrated in Figure 5-1.

$$\% \text{ deviation} = \frac{\text{experimental} - \text{model}}{\text{experimental}} \times 100 \quad (5-1)$$

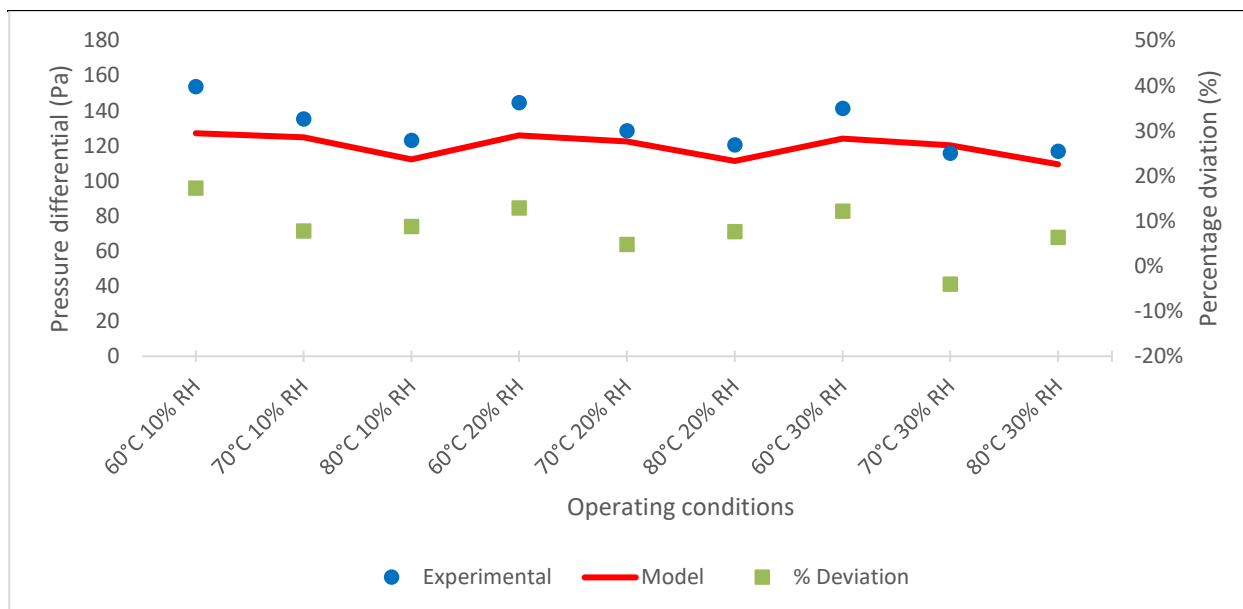


Figure 5-1: Ca(OH)_2 - Experimental and modelled pressure differential across the riser at different operating conditions

The blue dots originate from experimental data and the red line from modelled data, both represented on the primary y-axis. The green data points depicts the percentage deviation as determined by Equation 5-1 and is represented on the secondary y-axis. Each set of

corresponding blue, red and green data points represents a different operating condition including a specific temperature and relative humidity (RH) in the riser, as denoted by the x-axis. It is observed that the modelled pressure differential is slightly lower than the experimental values except for the 70°C and 30% relative humidity (RH) experiment. The largest deviation is seen for the 60°C and 10% RH, and the smallest deviation for the 70°C and 20% experiment. The percentage deviation ranges from -4.1% to 17.3%, with an average of 8.2% deviation. A deviation of 8.2% equates to approximately 10 Pa which is relatively small. From the graph it can be assumed that the model is capable of predicting the pressure differential across the riser for different operating conditions.

In Figure 5-2, the temperature at the inlet, T_1 , T_2 and T_3 are reported for both the experimental and modelling values. These respective temperature points reported are indicated on Figure 3-6 in Chapter 3.

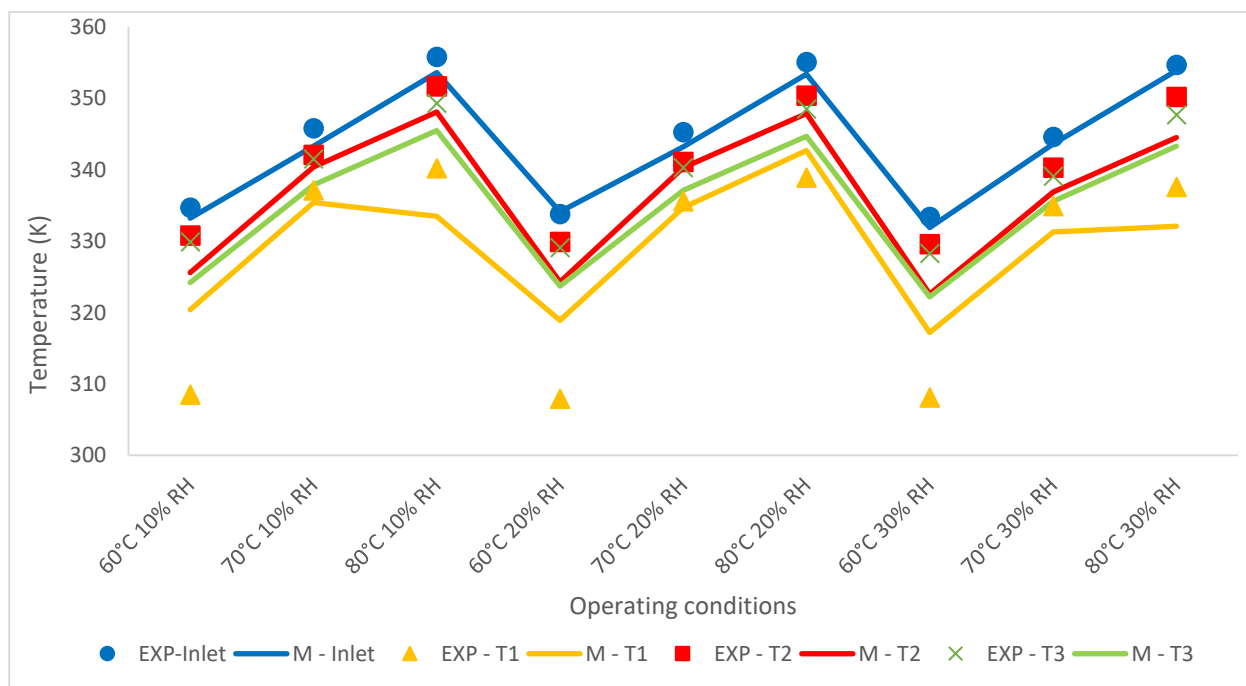


Figure 5-2: Ca(OH)_2 - Experimental and Modelled Temperature at four different point in the riser

The data points originate from experimental data, while the solid lines originate from modelled data. Both data sets are represented on the y-axis. Each set of data points, represents a different operating condition, while the different colours represent the different temperature points as identified earlier. The different operating conditions are presented on the x-axis.

It is easily observed, not only in exact value, but also in observed tendencies, that there exists a general good agreement between the different data sets where the largest deviation is observed for the temperature recorded at T_1 .

In conjunction with Figure 5-2, the percentage deviation of the modelled temperatures from the experimental values are reported in Table 5-2, together with the R-squared (R^2) value for each operating condition. The R^2 value, indicating the goodness-of-fit for two data sets, was determined by plotting the modelled data against the experimental data.

Table 5-2: Ca(OH)_2 – Percentage deviation of the modelled temperature from the experimental temperature and the R^2 -value for different operating conditions

Experimental/ Model Conditions	Percentage deviation (%)				R^2
	Inlet	T_1	T_2	T_3	
60°C 10%	0.4%	-3.9%	1.6%	1.7%	0.6631
70°C 10%	0.7%	0.5%	0.5%	1.1%	0.9367
80°C 10%	0.6%	2.0%	1.0%	1.1%	0.9994
60°C 20%	-0.1%	-3.6%	1.7%	1.6%	0.6149
70°C 20%	0.6%	0.3%	0.2%	0.9%	0.9195
80°C 20%	0.5%	-1.1%	0.7%	1.1%	0.8022
60°C 30%	0.4%	-3.0%	2.1%	1.9%	0.6495
70°C 30%	0.3%	1.1%	1.0%	1.0%	0.9831
80°C 30%	0.2%	1.6%	1.6%	1.3%	0.9646

The deviation in temperature is observed to be small for all experiments and temperature points. The largest deviation is observed at T_1 , for the 60°C and 10% RH conditions, the deviation for this condition is also negative which indicates that the modelled data is higher than the experimental data. Moreover, it is seen that temperatures at T_1 have the largest deviations. From the R^2 values, it is clearly observed that the agreement of experimental data and modelled data is high with most values exceeding a value of 0.9. The operating conditions for 60°C, however, show the lowest agreement, with 0.6631, 0.6149 and 0.6495 respectively. This deviation in data can be ascribed by the large deviation in temperatures at T_1 for operating conditions at 60°C.

The last consideration for the validation of the model is the solids distribution. In this case the radial solids distribution was considered. For validation purposes, only the bottom port and top port is illustrated in Figure 5-3 and Figure 5-4. Figure 5-3 and Figure 5-4 are so called parity plots for the modelled data against the experimental data, to predict the goodness-of-fit between the two data sets.

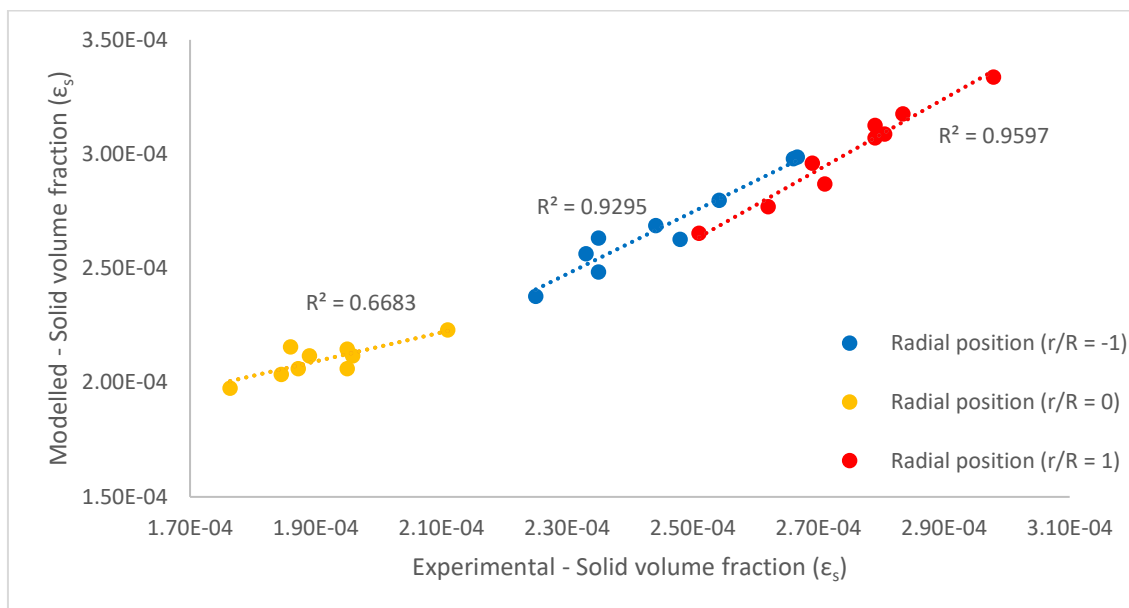


Figure 5-3: Ca(OH)_2 – Modelled vs experimental radial solids volume fraction at the bottom port of the riser

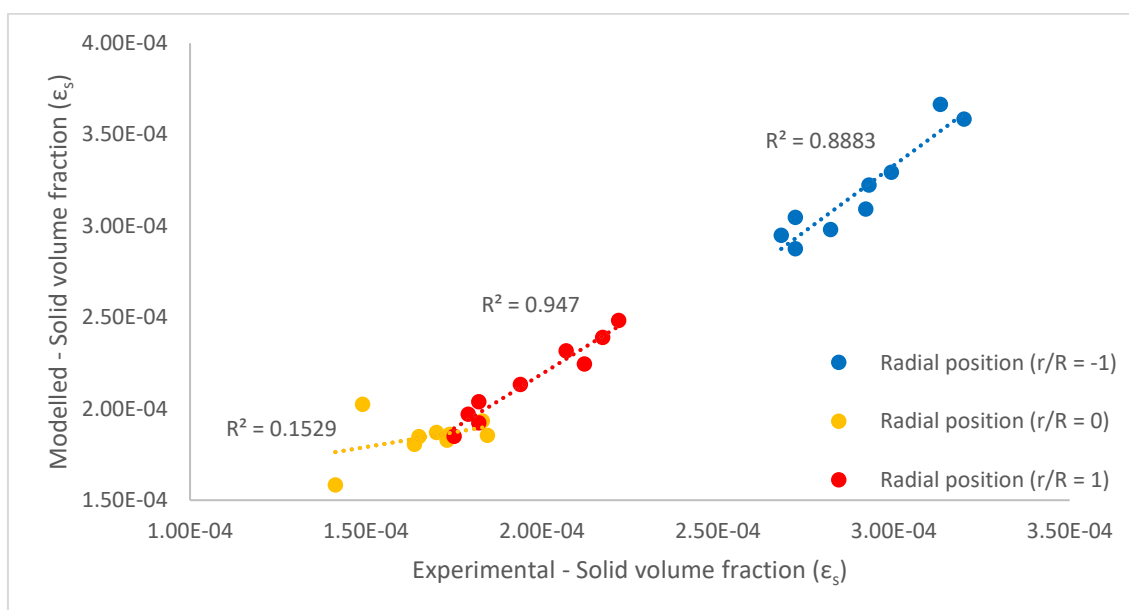


Figure 5-4: Ca(OH)_2 – Modelled vs experimental radial solids volume fraction at the top port of the riser

In both Figure 5-3 and Figure 5-4, the data points each has coordinates (x, y) where x represents the experimental data and y the modelled data for each of the respective operating conditions and is presented on the y-axis and x-axis respectively. The dashed lines are linear trend lines which represents the trend and resemblance between the data. Furthermore, the different radial positions as indicated in Figure 3-3, are each presented by a different colour.

For the bottom port, as illustrated in Figure 5-3, it is firstly observed that the solids volume fraction in the core or the riser ($r/R = 0$) is lower than near the walls. It is also in the core zone of the riser where the correlation between experimental and modelled solids volume fraction is the lowest

with a R^2 -value of 0.6683. The data near the wall of the riser shows satisfactory correlation with the R^2 -value of 0.9295 and 0.9597 respectively.

The top port, as presented in Figure 5-4, exhibits slightly different observations than the bottom port. The lowest solids volume fraction is still observed in the core zone of the riser, but here the solids volume fraction for $r/R = 1$ is closer to that of the core zone than that of the other riser wall at $r/R = -1$. The core zone of the riser again indicates the weakest correlation, but for the top port the correlation is extremely poor with a R^2 of 0.1529. This weak correlation can be mainly ascribed by two identified operating conditions which deviates significantly. These conditions are 80°C and 20% RH where both the experimental and modelled solids volume fraction were the lowest and 80°C and 10% RH. The deviation in data for these two conditions were calculated as -36% and -12% respectively. There is no obvious reason for this significant deviation. Furthermore, the correlation of data near the wall are satisfactory, with R^2 being 0.8883 and 0.947 respectively.

Despite having some deviations between the data, overall there exists adequate agreement between data for pressure differential, temperature and radial solids distribution. It can therefore be assumed that the model is sufficient for predicting the hydrodynamics of hydrated lime in a CFB riser.

5.2.2 Potassium Carbonate model validation

The discussion on the experimental and modelled potassium carbonate hydrodynamics will follow a similar order as with the hydrated lime discussed above.

In Figure 5-5, the experimental and modelled pressure differential values are plotted for each of the operating conditions. The percentage deviation of the modelled data from the experimental data is also reported in Figure 5-5.

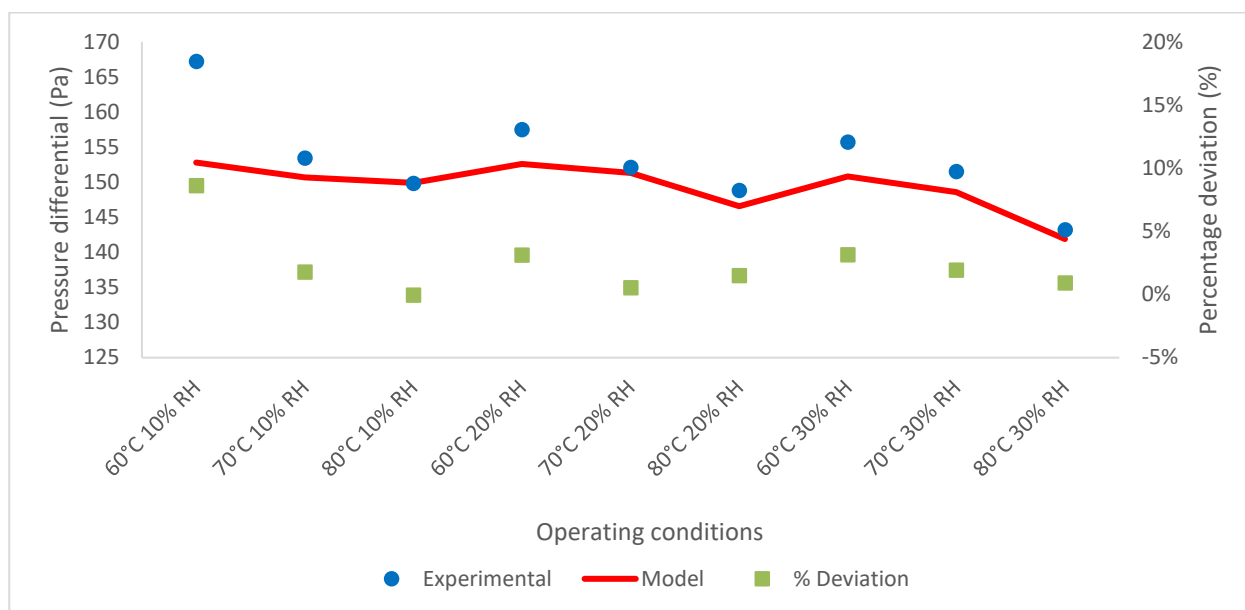


Figure 5-5: K_2CO_3 - Experimental and modelled pressure differential across the riser at different operating conditions

In Figure 5-5, the blue data points represent experimental data while the red line represents modelled data. The experimental and modelled data are presented on the primary y-axis, while the different operating conditions are presented on the x-axis. Furthermore, the percentage deviation is illustrated by the green data points and presented in the secondary y-axis.

It is observed that the overall trend of the data sets is very similar. Increasing the temperature while maintaining a fixed inlet gas relative humidity results in a decrease in pressure differential across the riser. For three of the operating conditions, 80°C 10% RH, 70°C 20% RH and 80°C 30% RH, the deviation between the data sets are significantly smaller when compared with the other conditions. These deviations correspond to -0.1%, 0.5 % and 0.9% respectively. The largest deviation is observed for the conditions of 60°C 10% RH and is calculated as 9%. However, the significantly large deviation cannot be ascribed to any particular reason and is just a stand-out observation to be made. Overall, the average deviation is approximately 2%. This indicates that the capability of the model to predict the pressure differential is very good.

For the temperature across the riser, Figure 5-6 illustrates the temperature at the inlet, T_1 , T_2 and T_3 , for both the experimental and modelling data. These respective temperature points reported are indicated on Figure 3-6 in Chapter 3.

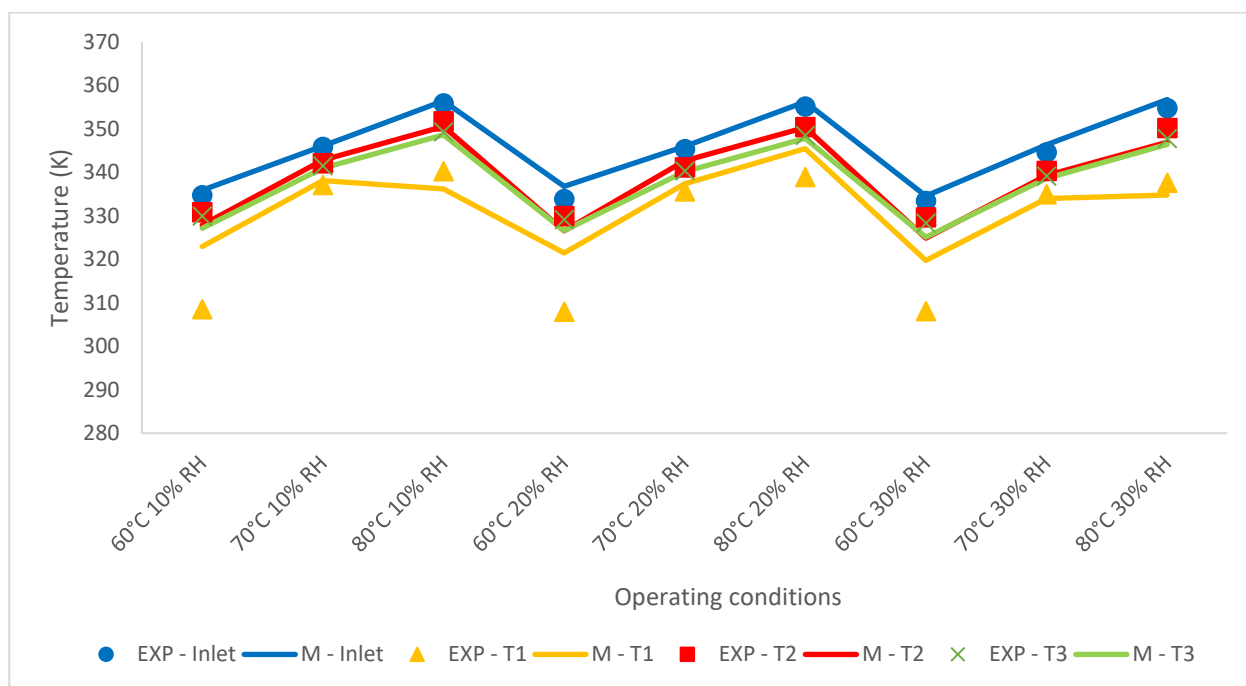


Figure 5-6: K_2CO_3 - Experimental and modelled Temperature at four different point in the riser

Figure 5-6 is illustrated in the same format as described earlier for hydrated lime. It is easily observed that overall, the tendencies depicted by the respective data sets are very similar. From these tendencies, the good agreement between the modelled and experimental temperature values for the inlet, T_2 and T_3 can be observed, which will be confirmed with the percentage deviation and R^2 -values reported in Table 5-3. It is further observed that the temperature modelled at T_1 shows the largest deviation from the experimental values.

In Table 5-3, the same observations could be made as the percentage deviation at T_1 has been calculated to be the highest for majority of the conditions. However, the largest deviation is -3.1% which is still relatively small. Overall, the data indicates a good agreement as most of the deviations are close to 0%. This then indicates that the model is sufficient for predicting the temperature throughout the riser.

Table 5-3: K_2CO_3 - Percentage deviation of the modelled temperature from the experimental temperature and the R^2 -value for different operating conditions

Experimental/ Model Conditions	Percentage deviation (%)				R^2
	Inlet	T_1	T_2	T_3	
60°C 10%	-0.3%	-3.1%	1.8%	1.9%	0.5147
70°C 10%	0.0%	1.2%	0.8%	1.2%	0.8047
80°C 10%	-0.1%	2.7%	1.3%	1.2%	0.9566
60°C 20%	-0.8%	-2.8%	2.0%	1.8%	0.4944

Experimental/ Model Conditions	Percentage deviation (%)				R ²
	Inlet	T ₁	T ₂	T ₃	
70°C 20%	-0.1%	1.0%	0.5%	1.1%	0.8631
80°C 20%	-0.3%	-0.4%	1.0%	1.2%	0.6661
60°C 30%	-0.3%	-2.2%	2.4%	2.0%	0.5266
70°C 30%	-0.5%	1.8%	1.3%	1.2%	0.8727
80°C 30%	-0.5%	2.3%	1.9%	1.4%	0.8704

The deviation in temperature is observed to be small with deviation ranging between -3.1% and 2.4%. The temperature at T₁ for the operating condition of 60°C and 10% RH corresponds to the deviation of -3.1%. The negative value indicated that the modelled data is higher than the experimental data. The 2.4% deviation corresponds to the conditions of 60°C and 30% RH. The temperatures at T₁ is also observed to have the largest deviation in temperatures between the data. Furthermore, the R²-value reported in the table is significantly lower for the operating condition of 60°C compared to the other temperatures. The values were 0.5174, 0.4944 and 0.5266 for a relative humidity of 10% RH, 20% RH and 30% RH respectively. In conjunction with the observations from Figure 5-6, the large deviations in temperatures recorded at T₁ for operating conditions of 60°C contributes to the lower agreement in data for this temperature. However, for the other operating conditions, most R²-values exceed 0.8, indicating a satisfactory agreement of the data.

In Figure 5-7, the radial solids volume fraction of experimental and modelled data of potassium carbonate at the bottom port is illustrated. Figure 5-8 presents the same data, but only for the top port. In both cases, the data is presented for all off the operating conditions.

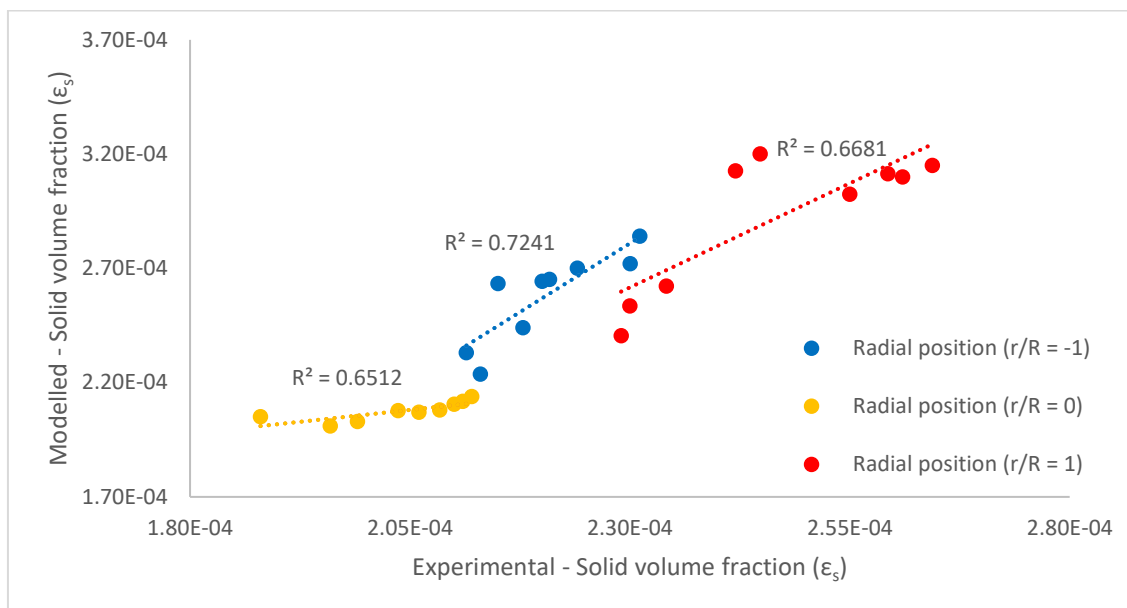


Figure 5-7: K_2CO_3 - Modelled vs experimental radial solids volume fraction at the bottom port of the riser

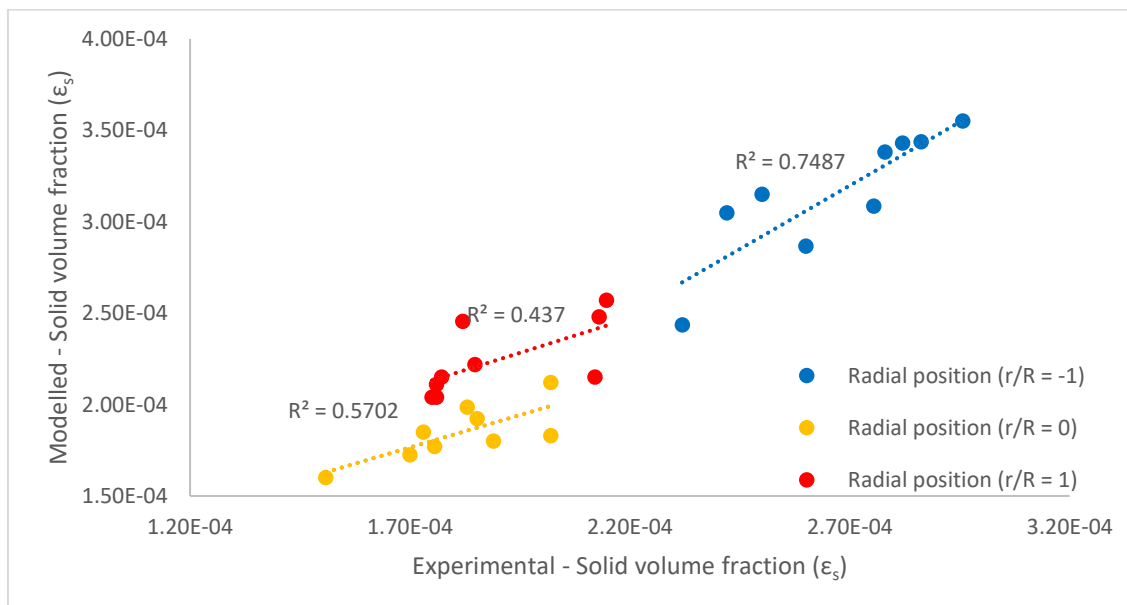


Figure 5-8: K_2CO_3 - Modelled vs experimental radial solids volume fraction at the bottom port of the riser

The same observations could be made when the data points at the top port in Figure 5-8 are investigated. The deviation is small. It can be seen that the modelled values are slightly higher than the experimental values, for both the bottom and top port. This can mainly be ascribed to the inability to capture downward flowing particles during the experiments which results in a lower solids volume fraction measured.

A first observation that can be made from Figure 5-7, is the clear difference in solids volume fractions for the respective radial points. It is seen that the core zone has the lowest solids volume fraction while the region close to riser walls has higher solids volume fractions. This is also an indication of asymmetric flow that is present in the riser. The correlation between experimental

and modelled data is also the lowest in the core zone and corresponds to a R^2 -value of 0.6512. The radial position $r/R = -1$ shows the highest correlation with a R^2 -value of 0.7241.

In Figure 5-8, a significant observation made is the solids volume fraction at the two radial points near the riser wall which are different from the bottom port. The lowest solids volume fraction is however, still observed at the core zone of the riser. In this case the lowest correlation is determined to be in one of the near-wall regions ($r/R = 1$) which is different from the bottom port. The correlation of this near-wall region correspond to a value of 0.467. Furthermore, the correlation in the core zone is 0.5702 and in the other near-wall region ($r/R = -1$) it is 0.784. The correlations are not significantly high but, deemed satisfactory, seeing that the riser exhibit chaotic flow, especially near the riser outlet, which can contribute to the deviations in data for the top port.

Despite having deviations between the modelled and experimental data, overall, there exist reasonable agreement between data for pressure differential, temperature and radial solids distribution. It is only in the case for solids distributions where significantly lower correlations has been observed. It can therefore be assumed that the model is sufficient for predicting the hydrodynamics of potassium carbonate in a CFB riser.

5.3 Influence of operating parameters

5.3.1 Hydrated Lime

5.3.1.1 Influence of inlet gas temperature

(a) Pressure

Pressure is an important parameter in the description of the hydrodynamics and thus the operation of a CFB. To investigate the influence of the inlet temperature and inlet relative humidity on the operation of the riser, the pressure differential across the entire length of the riser, as well as across the upper region of the riser as defined in Figure 3-6 of Chapter 3.4.4, had to be considered. The pressure differential experimental results are reported in Table 5-4.

Table 5-4: Ca(OH)_2 - Experimental pressure differential across (i) the riser and (ii) the riser upper region under varying inlet gas conditions

Inlet gas relative humidity (%RH)	Inlet gas temperature (°C)	ΔP_{riser} (Pa)	$\Delta P_{\text{upper region}}$ (Pa)
10	60	153.6	12.2
	70	135.2	10.8

Inlet gas relative humidity (%RH)	Inlet gas temperature (°C)	ΔP_{riser} (Pa)	$\Delta P_{upper\ region}$ (Pa)
20	80	122.9	9.8
	60	144.5	11.5
	70	128.5	10.2
	80	120.5	9.6
	60	141.3	9.3
30	70	115.6	9.2
	80	116.8	6.9

Looking at the experimental results, it is observed that when the inlet gas relative humidity is kept constant, and the temperature is increased from 60°C to 80°C, the pressure differential ultimately decreases across the riser, as well as across the upper section of the riser. This observation can mainly be ascribed to the changes in solids holdup when the inlet temperature is increased. This will be discussed in more detail in Section 5.3.1.1 (c). An increase in inlet gas temperature also results in the density of the gas being decreased and the gas viscosity being increased, and hence the magnitude of drag force is changed. As the gas temperature is increased and the gas density decreases, the gas consequently exerts less pressure contributing to the drop in pressure. When excluding the effect of the venturi, the pressure differential is much lower across the upper region compared to the riser. The pressure differential across the upper region of the riser can be expressed as approximately 8% of the total pressure that is lost across the riser.

In Figure 5-9, the general static pressure profile across the riser is presented as retrieved from modelling. The areas of interest, that corresponds to Figure 3-6 are labelled (a), (b) and (c) respectively and reported in Table 5-5. These areas represent the riser inlet, conical shaped area and top of the riser respectively.

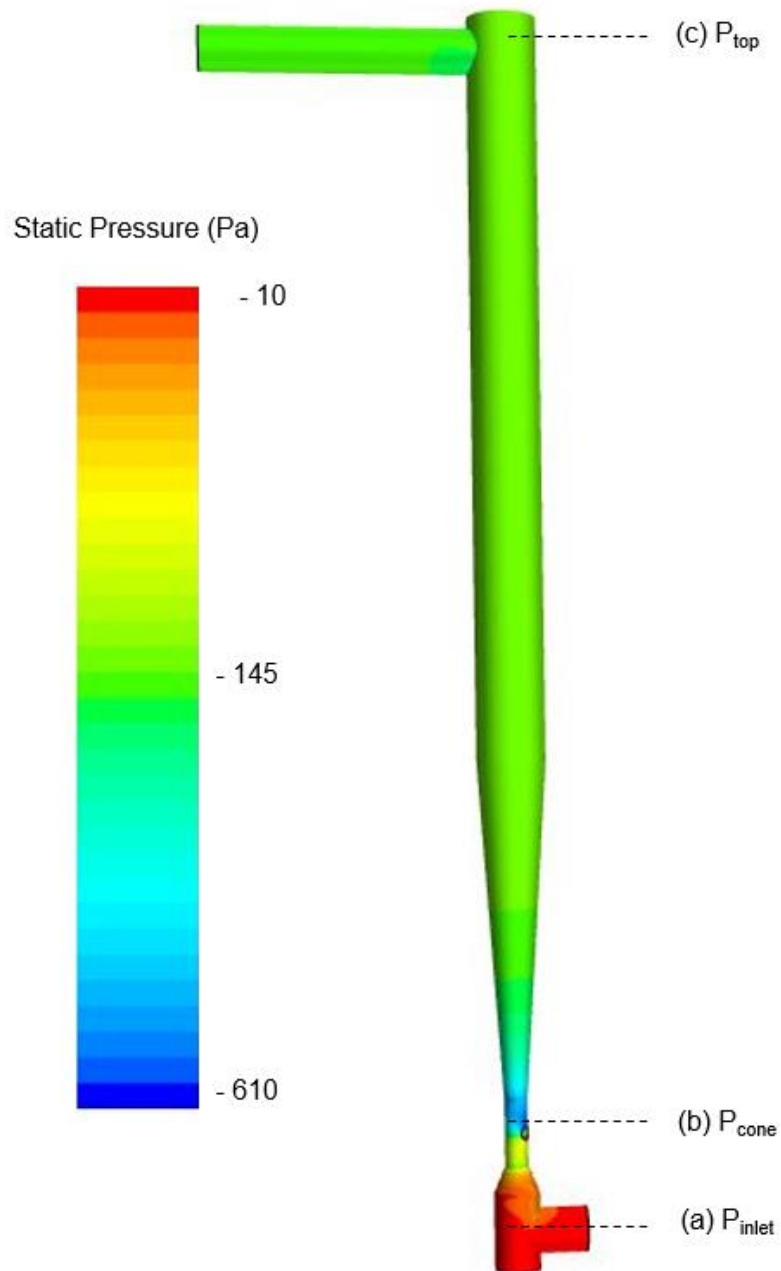


Figure 5-9: Ca(OH)_2 - General static pressure profile across the riser

The modelled pressure at the top of the riser was subtracted from the modelled inlet pressure to calculate the pressure differential across the riser. The pressure differential is also provided in Table 5-5.

Table 5-5: Ca(OH)_2 - Modelled static gauge pressure values for positions (a), (b) and (c) and modelled pressure differential across the riser

Inlet gas relative humidity (%RH)	Inlet gas temperature (°C)	(a): P_{inlet} (Pa)	(b): P_{cone} (Pa)	(c): P_{top} (Pa)	ΔP_{riser} (Pa)
10	60	-9.04	-595.8	-136.1	127.1
	70	-12.6	-597.3	-137.3	124.7
	80	-27.5	-599.4	-139.7	112.2
20	60	-10.8	-602.1	-136.7	125.9
	70	-15.7	-604.7	-138.1	122.4
	80	-30	-605.6	-141.3	111.3
30	60	-13.8	-607.4	-137.9	124.1
	70	-19.4	-609.1	-139.7	120.3
	80	-32.7	-609.7	-142.1	109.4

It is observed that in the bottom of the riser the modelled pressure was close to atmospheric pressure, corresponding to the experimental results. A distinctive observation is the sudden drop in pressure in the riser conical area for which values are given in the 4th column of Table 5-5. The sudden drop in pressure is the result of the inlet of solids from the side of the venturi throat in combination with the restriction in flow in the riser geometry. The restriction causes acceleration of the flow resulting in low static pressure observed after the venturi section of the riser.

Again, the same tendency can be seen as with the experimental results. An increase in temperature, while all other variables are kept constant, results in a decrease in pressure at all identified points as well as the pressure differential across the riser. However, the greatest portion of pressure losses occur around the venturi and overshadow the pressure loss across the upper region.

(b) Temperature

The temperature inside the riser is very important in the operation of a CFB, especially when reactions take place. It is necessary to have a uniform temperature distribution for optimal process conditions. The experimental temperature results are reported in Table 5-6, at four respective points in the riser, being the temperature at the inlet, and three points in the upper region of the riser. The different measurement points correspond to the points as shown in Figure 3-6.

Table 5-6: Ca(OH)_2 - Experimental temperatures at different points across the riser for varying inlet gas temperature

Inlet gas relative humidity (%RH)	Inlet gas temperature ($^{\circ}\text{C}$)	T_{inlet} (K)	T_1 (K)	T_2 (K)	T_3 (K)
10	60	334.7	308.5	330.8	329.9
	70	345.8	337.1	342.1	341.5
	80	355.8	340.2	351.7	349.3
20	60	333.8	307.9	329.9	329.1
	70	345.3	335.6	341.1	340.3
	80	355.1	338.9	350.4	348.5
30	60	333.4	308.1	329.6	328.3
	70	344.6	334.9	340.3	339.1
	80	354.7	337.6	350.2	347.7

During the experimental runs, it was necessary for the inlet temperatures to be as close as possible to the desired inlet gas temperature, which was done successfully. The observation made from this data is that the inlet gas temperature, will be the determining factor of the temperature conditions inside the riser. As the inlet gas temperature increases, the temperature in T_1 , T_2 and T_3 increases with approximately the same orders of magnitude. It is seen that the temperature recorded at T_1 is relatively lower than those measurements at the other points. The reason for this is the solids being introduced between the inlet and T_1 . It was however not possible to record the temperature within this region during the experiments, but it will be addressed during the discussion of modelled results. Furthermore, the temperature exhibits a uniform distribution, since the temperature loss across the riser ranges between 4 K and 7 K.

In Figure 5-10, the general temperature profile from the simulations are shown, indicating the different points corresponding to the experimental measurement positions, with exclusion of a measurement point in the conical area of the riser. The temperatures are presented in Table 5-7.

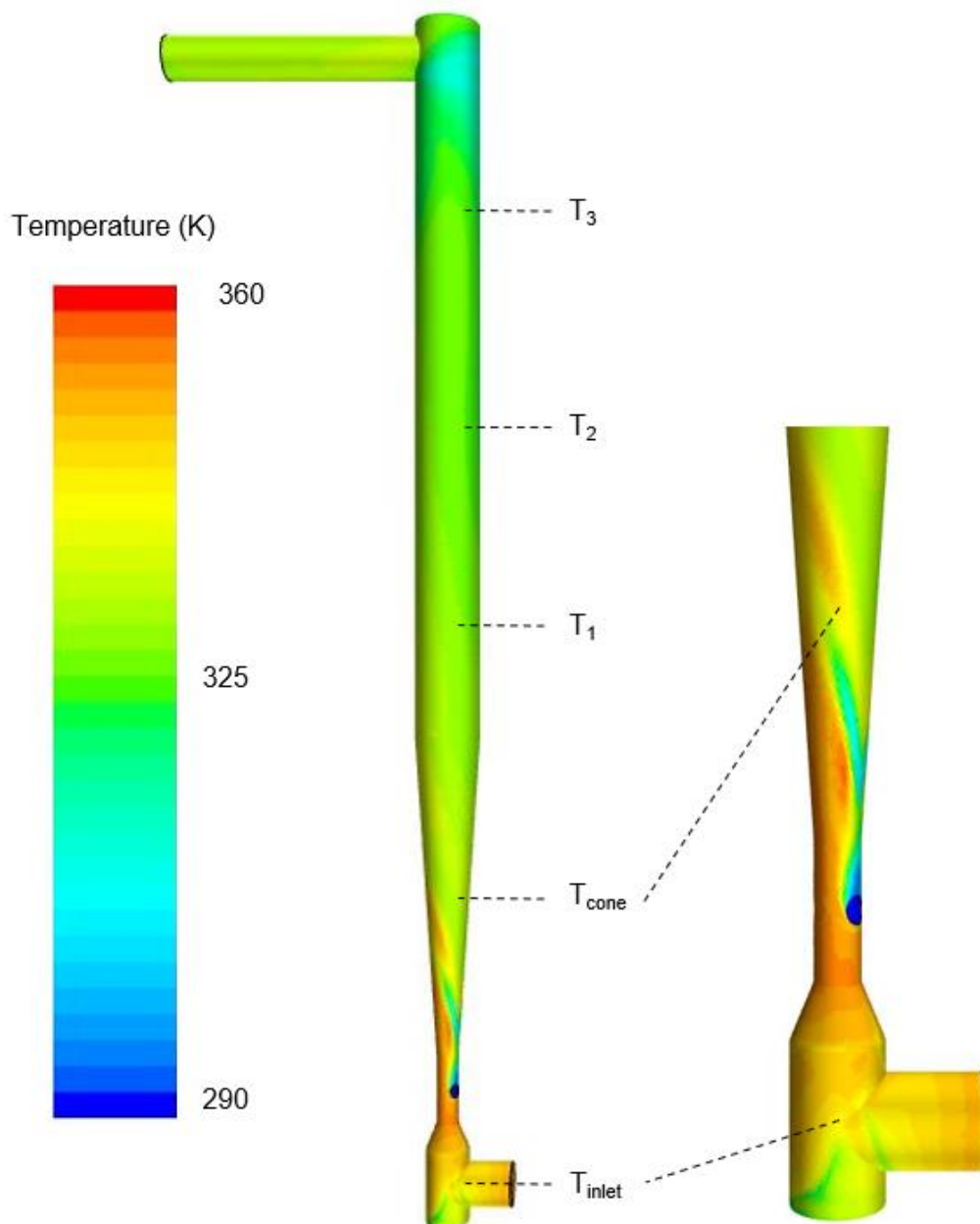


Figure 5-10: Ca(OH)_2 - General temperature profile of the riser for hydrated lime indicating the different temperature points

Table 5-7: Ca(OH)_2 - Modelled temperatures at different points across the riser for varying inlet gas temperature

Inlet gas relative humidity (%RH)	Inlet gas temperature ($^{\circ}\text{C}$)	T_{inlet} (K)	T_{cone} (K)	T_1 (K)	T_2 (K)	T_3 (K)
10	60	333.2	303.7	320.4	325.6	324.2
	70	343.3	313.9	335.4	340.4	337.9
	80	353.6	321.8	333.5	348.1	345.5

Inlet gas relative humidity (%RH)	Inlet gas temperature (°C)	T_{inlet} (K)	T_{cone} (K)	T_1 (K)	T_2 (K)	T_3 (K)
20	60	334.1	302.6	318.9	324.3	323.7
	70	343.2	313.7	334.7	340.3	337.1
	80	353.4	321.5	342.7	347.9	344.7
30	60	331.9	301.1	317.2	322.6	322.2
	70	343.6	310.7	331.3	336.9	335.6
	80	353.9	318.6	332.1	344.5	343.3

The presented data makes it clear that, the inlet gas temperature determine the temperature inside the riser. It is seen that the order in which the inlet temperature is increased is also applicable to the increases recorded in temperature axially along the riser. Another observation that can be made in the modelled data is the lower temperatures recorded in the conical area. This is mainly a result of the solids entering the riser just before the conical area. The solids are kept in a hopper exposed to the surrounding environment and upon entering the riser, the temperature drop can be ascribed to heat transfer from the warm gas to the solid particles. However, as the flow develop further along the riser, the temperature returns closely enough to the inlet gas temperature as specified. The difference in temperature at the top compared to the inlet temperature range from 6 K to 11K. The observations made from the modelling data agrees with the experimental observations made earlier.

(c) Solids distribution

The local solids volume fraction was recorded at three radial positions across the diameter of the riser, and at the three sampling ports, as discussed in Section 3.4.4. For this section, only the results pertaining to 10 % RH with varying inlet gas temperature conditions will be presented. The remainder of the results can be found in Annexure D and presents the same trends and observations as the results that will be discussed in this section.

In Figure 5-11 the radial profile for solids volume fraction at different axial positions in the riser is illustrated.

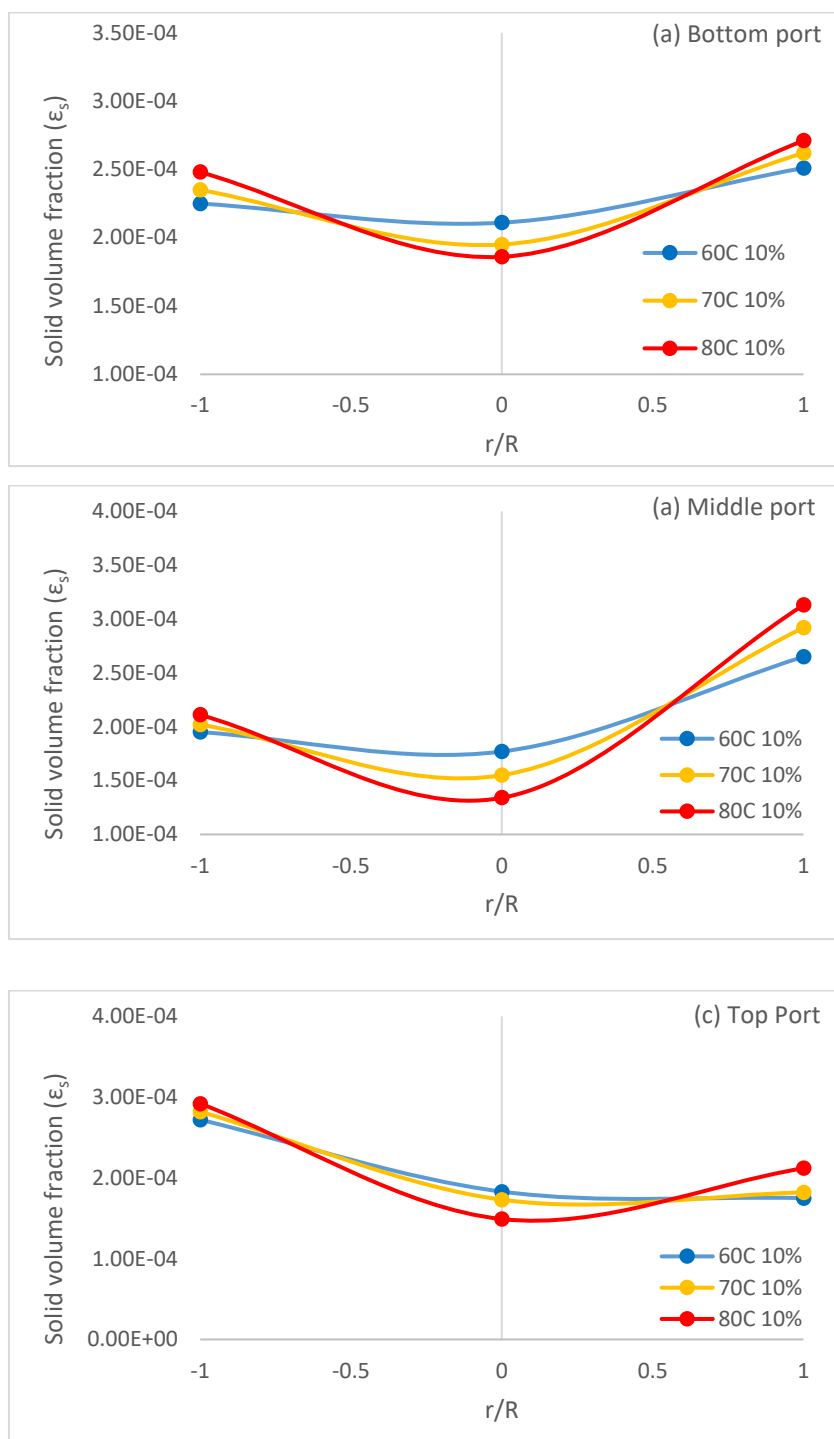


Figure 5-11: Ca(OH)_2 - Radial solids distribution profile at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas temperature

In Figure 5-11, the solids volume fraction is presented on the y-axis, with the radial position on the x-axis. The different colours represent the different operating conditions. To discuss the overall trend of the radial solids distribution in the riser, the top view of the riser as given in Figure 3-3 in conjunction with Figure 3-4 will be used for reference.

The gas is being injected from the reference point at 0° and the solids at approximately 30° . The riser exit is located at a position approximately 135° from the air inlet. The location of the entrance points, and momentum of the air flow and solid particles results in the solid particles being transported along the wall opposite to the solids inlet for a large part of the riser. This would explain the increased solid volume fraction on the one side of the riser which is slightly noticeable in the bottom and middle port radial profiles. The radial profiles exhibit a slight asymmetrical profile to the one side of a riser, while the higher concentration of seems to migrate to the other side around the top port. This can be as a result of the inlet-outlet configuration which will be further explained with the modelling results. Furthermore, the profiles exhibit a rather flat profile throughout, with moderate gradients between the solids volume fraction at the three radial points.

Although not prominent, there do exist a slight decrease in solids concentration in the core zone of the riser at all three sampling ports, with increasing temperature which corresponds to findings in literature. It seems that the particles tend to accumulate more near the wall with an increase in temperature from 60°C to 80°C . It is, however, not so evident and could be due to the temperature range being too small, as well as the low solids flux that was used for experiments. This effect will most likely be more noticeable when temperature differences are slightly larger.

It is evident that the typical core-annulus flow as described in Chapter 2.2.2 cannot be identified within this system, considering the experimental values reported in Figure 5-11. This can be ascribed to the combination of chosen velocity, the particles which are classified as Geldart Group C and the low solids flux used in the experiments.

The radial solids distribution from the CFD simulations at the three different ports are illustrated in Figure 5-12. The geometry of the riser was cut into different plane sections for ease of comparing the results with the experimental results as illustrated in Figure 5-11.

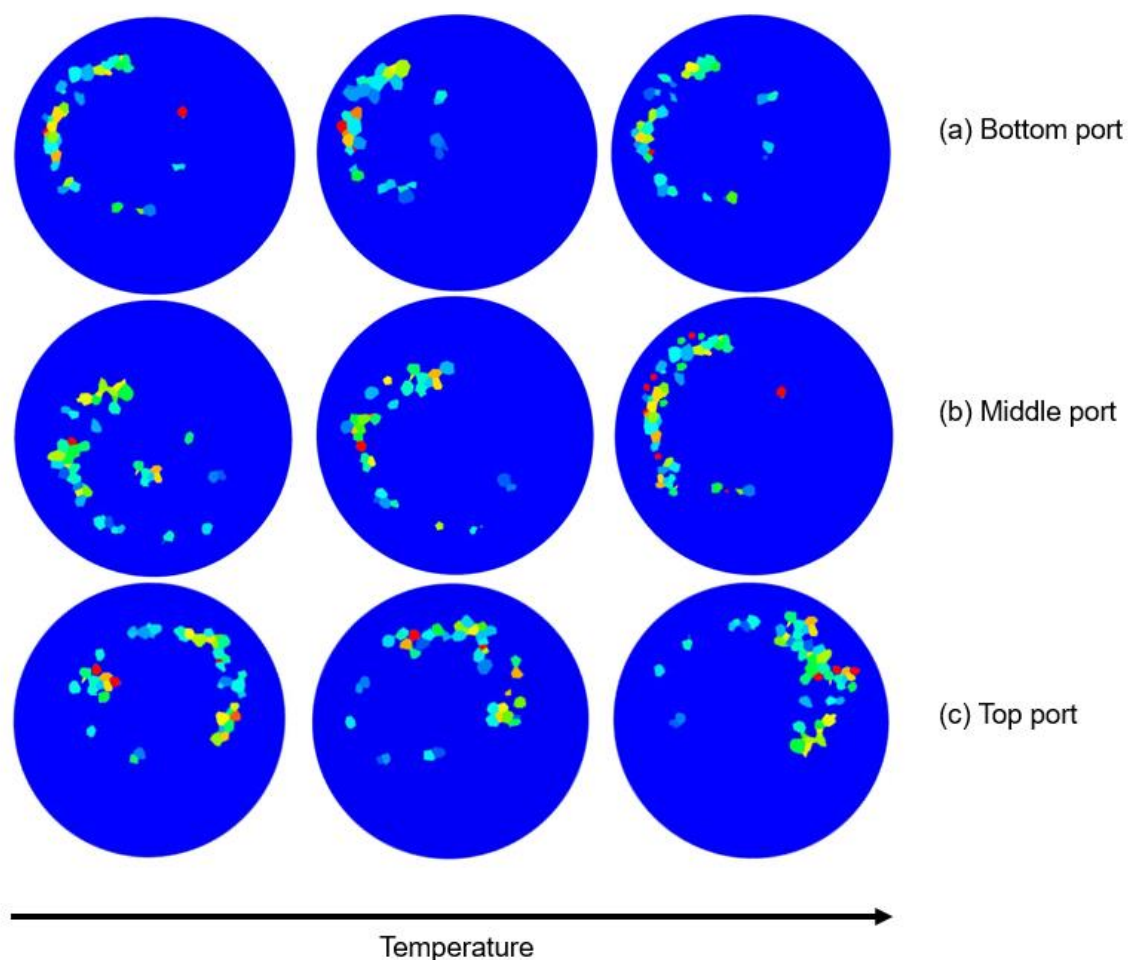


Figure 5-12: Ca(OH)_2 – Modelled radial solids volume fractions at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas temperature

The profiles are organised in such a manner that temperature increases from left to right, and with the bottom port in the top row, middle port in the middle row, and top port in the bottom row. The illustrated figures have a top view, with its orientation the same as the reference image in Chapter 3.2.1. The findings as discussed earlier are now much more evident, and easily identified. In the CFD simulation the particles are grouped together in parcels to ease computation, and each dot represents a cluster of similar particles.

When considering Figure 5-12, with an increase in temperature (left to right), the percentage of area covered by particles in the core region of the riser is slightly reduced and the area covered along the walls slightly increases. This agrees well with the experimental observations as reported in Figure 5-11.

Figure 5-13 displays the average axial profile of the solids volume fraction for varying inlet gas temperatures at 10 %RH. The axial distribution is displayed only for the upper region of the riser after the conical area, with the reference point at height = 0m, as identified in Figure 3-6 of Chapter

3.4.4. The average axial volume fraction ($\bar{\epsilon}_s$) was calculated by considering an integral method which considers the local solid volume fraction calculations from sampling measurements.

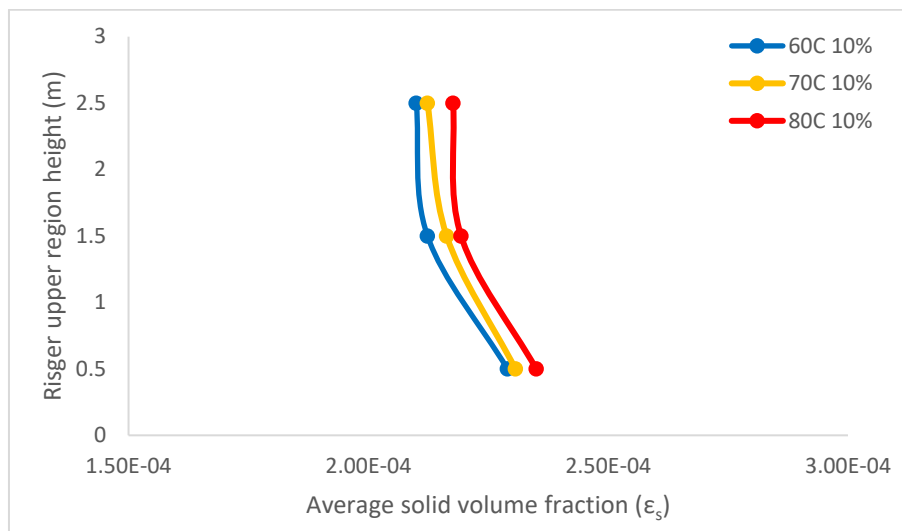


Figure 5-13: Ca(OH)_2 - Experimental average axial solids distribution for varying inlet gas temperature

It is observed that for a constant inlet gas relative humidity, the axial distribution for the different temperatures are very similar. The solids volume fraction also tends to be relatively constant in the upper half of the riser, as illustrated by Figure 5-13, where the difference in top and bottom average solid volume fraction ranges from 1.7×10^{-5} to 1.9×10^{-5} as the inlet gas temperature increases. The average solids volume fraction is also seen to increase with increasing temperature for both the experimental and modelling results. The observations made from the axial distribution in the upper region of the riser are in accordance with findings by Zhang, 2018.

In the CFD environment, the circular planes that were created along the height of the riser were utilized to extract the surface average solids volume fraction from the simulations. The profiles that were created from the extracted data for the operating conditions of varying inlet gas temperature and 10% RH, are illustrated in Figure 5-14. The profiles of other operating conditions are presented in Annexure F since the same trends were identified as what will be discussed here.

Again, the axial profile y-axis reference point is the same as specified during the experimental discussion. However, during simulations the bottom region of the riser was accessible and the average solids volume fraction at various points will be included in the profiles.

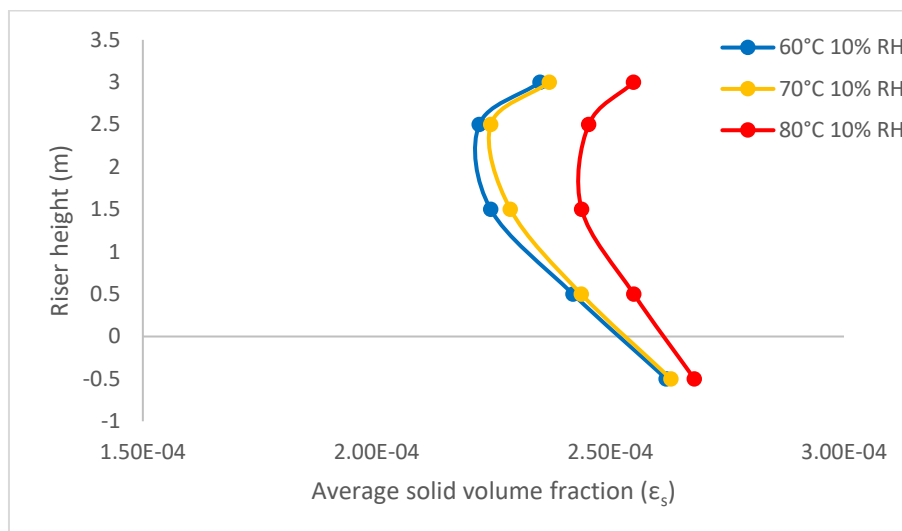


Figure 5-14: Ca(OH)_2 - Modelled average axial solids distribution for varying inlet gas temperature

The profiles at all of the different temperatures exhibit a general C-shaped curve. It is seen that in the bottom region the concentration of solids decreases along the riser height until approximately at the top of the conical section. Although the C-shape profile is commonly reported in literature as a result of flow development, in this study it is thought to be as a result of the increasing pipe diameter of the cone. The increasing concentration of solids at the top of the riser was likely caused by the T-shape exit and particles colliding with the top of the riser. This agrees with the findings from the radial distributions as well. From the axial distribution, it can be seen that the average solids volume fraction is very similar for all the temperatures. The solids volume fraction also tends to be relatively constant in the upper region of the riser where the flow is more developed.

5.3.1.2 Influence of inlet gas relative humidity

The discussion for the influence of varying inlet gas relative humidity will have the same structure as with the inlet gas temperature. The figures and tables are also constructed in the same format.

(a) Pressure

To investigate the influence of varying inlet gas relative humidity, while the inlet gas temperature is kept constant, the pressure differential across the riser and across the upper region was considered. The experimental pressure differential results are reported in Table 5-8.

Table 5-8: Ca(OH)_2 - Experimental pressure differential across riser and riser upper region under varying inlet gas conditions

Inlet gas temperature ($^{\circ}\text{C}$)	Inlet gas relative humidity (%RH)	ΔP_{riser} (Pa)	$\Delta P_{\text{upper region}}$ (Pa)
60	10	153.6	12.2
	20	144.5	11.5
	30	141.3	9.3
70	10	135.2	10.8
	20	128.5	10.2
	30	115.6	9.2
80	10	122.9	9.8
	20	120.5	9.6
	30	116.8	6.9

It is clearly seen that increasing the inlet gas relative humidity results in a decrease in pressure differential across the entire riser as well as the upper region of the riser. Increasing the inlet gas relative humidity while the temperature stays constant, results in the water vapour fraction present in the gas to increase. As a result, due to the lower weight of water compared to air, the density of the air decreases and the pressure is decreased accordingly. This would explain the decrease in pressure differential. Furthermore, it is also observed that the majority of pressure losses occurs in the lower region of the riser where the venturi is present and significantly influences the pressure drop. The pressure differential in the upper region can be expressed as approximately 8% of the total pressure losses, with 92% of pressure losses in the lower region of the riser.

The previous general static pressure profile as given in Figure 5-9 in Section 5.3.1.1 (a) together with Figure 3-6, indicates the temperature points that is reported in Table 5-9. The modelled pressure differential across the entire riser are also reported in the last column of Table 5-9.

Table 5-9: Ca(OH)_2 - Modelled static gauge pressure values for positions (a), (b) and (c) on Figure 5.9 and modelled pressure differential across the riser

Inlet gas temperature ($^{\circ}\text{C}$)	Inlet gas relative humidity (%RH)	(a): P_{inlet} (Pa)	(b): P_{cone} (Pa)	(c): P_{top} (Pa)	ΔP_{riser} (Pa)
60	10	-9.04	-595.8	-136.1	127.1
	20	-10.8	-602.1	-136.7	125.9

Inlet gas temperature (°C)	Inlet gas relative humidity (%RH)	(a): P_{inlet} (Pa)	(b): P_{cone} (Pa)	(c): P_{top} (Pa)	ΔP_{riser} (Pa)
70	30	-13.8	-607.4	-137.9	124.1
	10	-12.6	-597.3	-137.3	124.7
	20	-15.7	-604.7	-138.1	122.4
	30	-19.4	-609.1	-139.7	120.3
80	10	-27.5	-599.4	-139.7	112.2
	20	-30	-605.6	-141.3	111.3
	30	-32.7	-609.7	-142.1	109.4

The reported values from varying the inlet gas relative humidity, indicates clearly that the pressure in all areas decreases as the relative humidity is increased. A distinctive observation is made by the significantly small pressure values in the conical area compared to the other. The sudden drop in pressure, also seen in the previous discussions, is mainly the result of the venturi effect. The modelled pressure and pressure differentials indicates the same findings as with the experimental results, however, there is a slight under prediction in the values as already explained in Section 5.2.

(b) Temperature

The influence of varying inlet gas relative humidity on the temperature in the riser are reported in Table 5-10. The experimental results consisted of the same four points along the riser that were discussed previously.

Table 5-10: Ca(OH)₂ - Experimental temperatures at different points across the riser for varying inlet gas conditions

Inlet gas temperature (°C)	Inlet gas relative humidity (%RH)	T_{inlet} (K)	T_1 (K)	T_2 (K)	T_3 (K)
60	10	334.7	308.5	330.8	329.9
	20	333.8	307.9	329.9	329.1
	30	333.4	308.1	329.6	328.3
70	10	345.8	337.1	342.1	341.5
	20	345.3	335.6	341.1	340.3
	30	344.6	334.9	340.3	339.1

Inlet gas temperature (°C)	Inlet gas relative humidity (%RH)	T_{inlet} (K)	T_1 (K)	T_2 (K)	T_3 (K)
80	10	355.8	340.2	351.7	349.3
	20	355.1	338.9	350.4	348.5
	30	354.7	337.6	350.2	347.7

A general observation can be made that while keeping the inlet gas temperature constant, and increasing the gas relative humidity, there exist a slight decrease in temperature in all areas. The decrease is in most cases so small that it is only with a fraction of 1 Kelvin. This observation agrees with general findings in the research field, however the combination of temperature and relative humidity used in this study can be the result for the significantly small changes.

The temperature values as predicted by the simulations are reported in Table 5-11. The modelled results show similar trends as with the experimental values.

Table 5-11: Ca(OH)_2 - Modelled temperatures at different points across the riser for varying inlet gas conditions

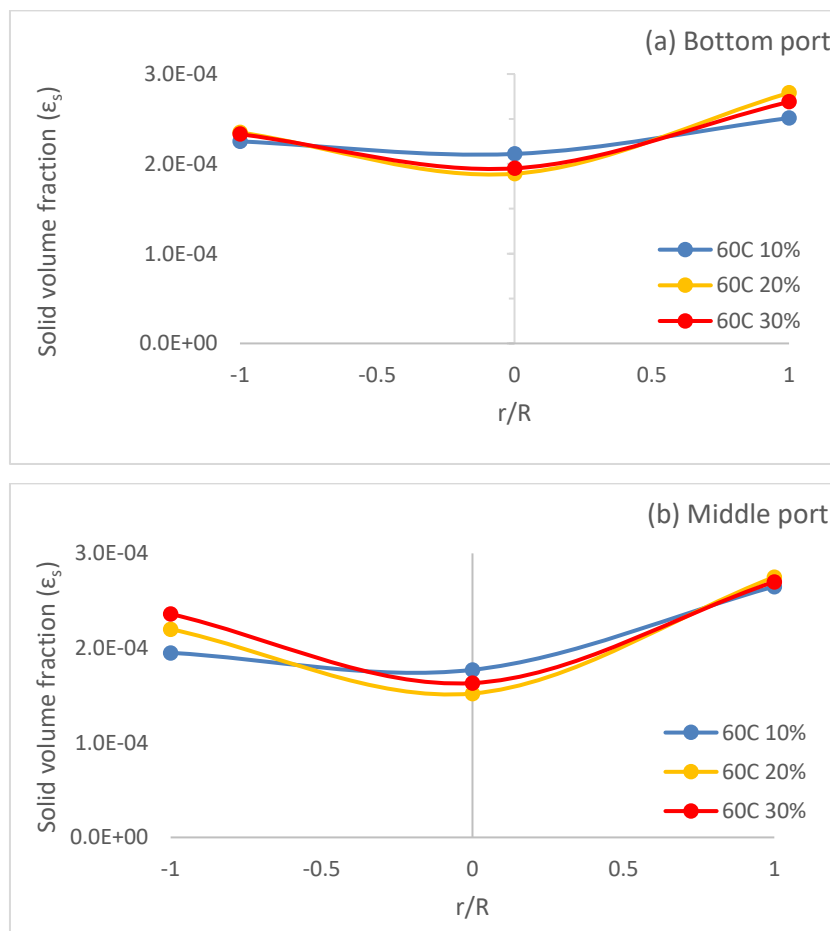
Inlet gas temperature (°C)	Inlet gas relative humidity (%RH)	T_{inlet} (K)	T_{cone} (K)	T_1 (K)	T_2 (K)	T_3 (K)
60	10	333.2	303.7	320.4	325.6	324.2
	20	334.1	302.6	318.9	324.3	323.7
	30	331.9	301.1	317.2	322.6	322.2
70	10	343.3	313.9	335.4	340.4	337.9
	20	343.2	313.7	334.7	340.3	337.1
	30	343.6	310.7	331.3	336.9	335.6
80	10	353.6	321.8	333.5	348.1	345.5
	20	353.4	321.5	342.7	347.9	344.7
	30	353.9	318.6	332.1	344.5	343.3

The temperatures throughout the riser slightly decreases while increasing the gas relative humidity. In the Table 5-11, the values for temperature in the conical area are reported and it can be seen that there exist a sudden drop in temperature from the inlet to the cone area. This is as a result of the solids particles being fed from ambient conditions, and heat transfer taking place. However, as the flow develops the temperature increases and stabilises in the upper region of the riser. The effect of varying inlet gas relative humidity on the temperature can be seen from

the data, but as a result from the combination of temperatures and relative humidity it is not as apparent.

(c) Solids distribution

In Figure 5-15 the radial solids distribution is presented for the operating conditions of 60°C and varying inlet gas relative humidity. Only the result of this specific condition is presented as the same trends could be seen in all other data than what will be explained here. The other experimental radial solids distribution data for varying inlet gas relative humidity can be seen in Annexure D.



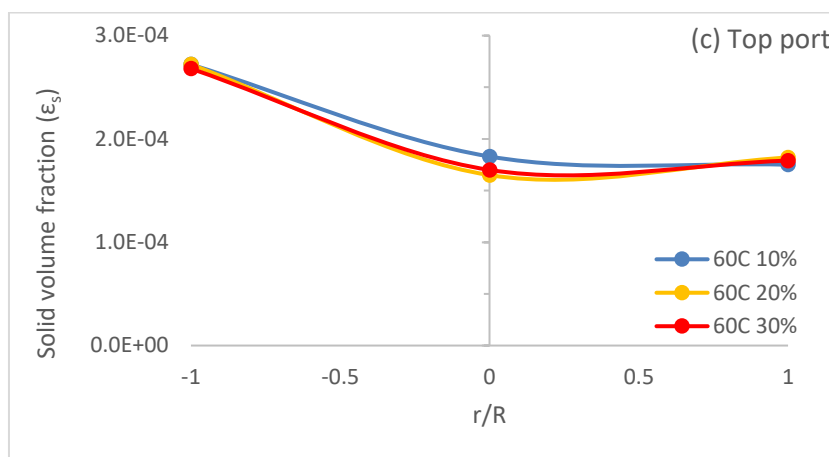


Figure 5-15: Ca(OH)_2 – Experimental radial solids distribution profile at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet conditions

In Figure 5-15, the solids volume fraction is presented on the y-axis, with the radial position on the x-axis where 0 indicate the centre of the riser. Similar tendencies are seen as with varying inlet gas temperature. Figure 3-3 and Figure 3-4 are used for reference. The solids are seen to preferably be transported against the opposite wall form the solids inlet, as a result of the solids momentum and air flow. Both the air inlet and solids inlet do play a large role in this observation. This would also explain the slight increased solids volume fraction to the one side of the riser. The outlet has a similar effect on the flow distribution, by redirecting the solids from the one side to the other in the top part of the riser, as seen previously. Furthermore, the radial distribution exhibits a rather flat distribution profile in both the bottom and middle port region, while the top port displays a larger concentration gradient to the one side of the riser. This can be largely ascribed to the T-shaped outlet and the chaotic flow in the top of the riser when particles collide with the top of the riser.

It is seen that the concentration of solids in the core zone of the riser slightly decreases with a gas relative humidity increase from 10% to 20% but increases slightly when the gas relative humidity is further increased to 30%. The concentration of solids near the wall region increase with an increase from 10% to 20%, while a slight decrease from 20% to 30% can be observed. This phenomenon can be as a result of the particle-particle and particle-wall electrostatic forces that decreases with an increase in gas relative humidity. For that reason, as the RH increases from 10% to 20%, the tendency of the hydrated lime particles to form blobs at the core and clusters at the riser wall is weakened. The hydrated lime particles now tend to be easily fluidised in particle groups in the upward direction.

The CFD simulations illustrates the above discussed trends even more. In Figure 5-16, the radial distribution at the three riser ports can be seen as a cross-sectional illustration. The profiles increase in inlet gas relative humidity from left to right, while the bottom port is at the top, and the

top port is in the bottom row. The changes in solids concentration for the core zone and wall region of the riser can be seen very clearly and support the findings from the experimental results.

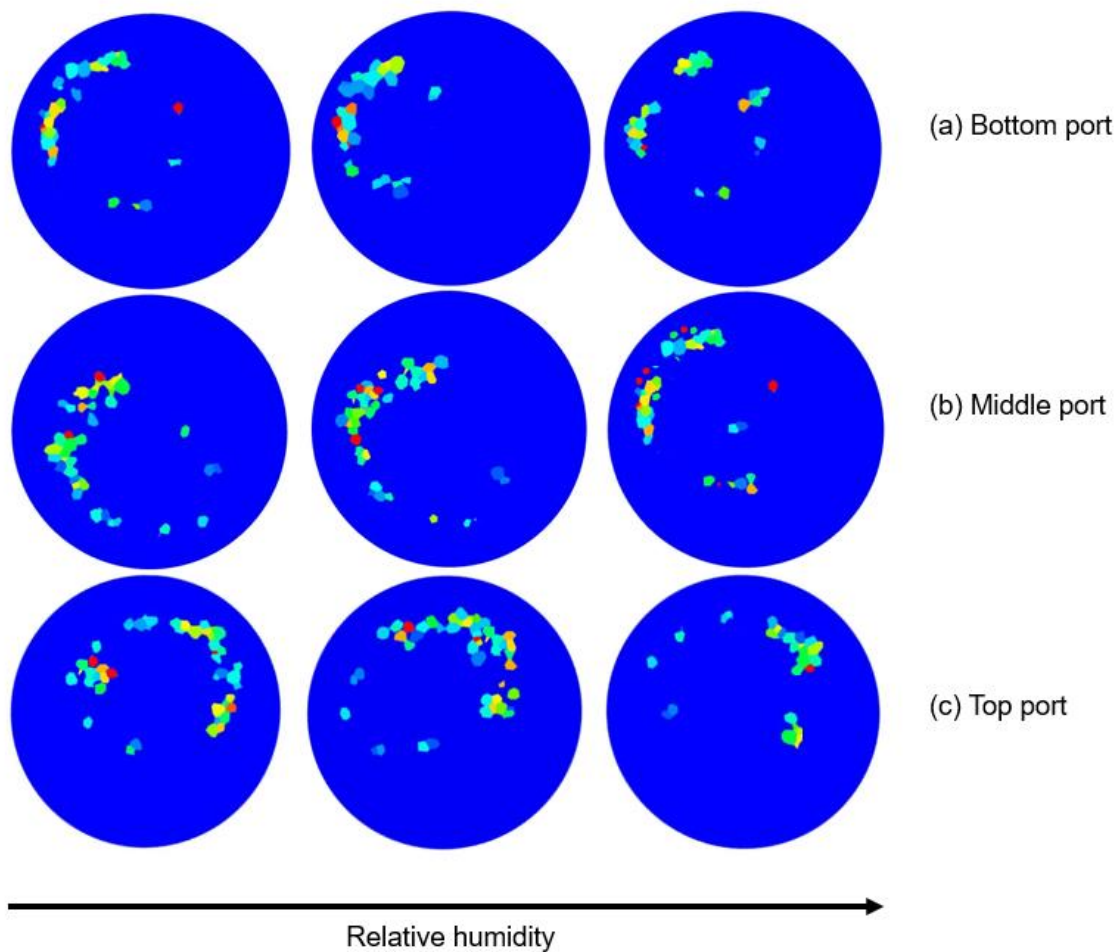


Figure 5-16: Ca(OH)_2 - Modelled radial solids volume fractions at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas conditions

Figure 5-17 displays the average axial distribution profile of the solids volume fraction for varying inlet gas relative humidity for 60°C . The axial distribution is displayed only for the upper region of the riser after the cone as identified in Chapter 3.4.4.

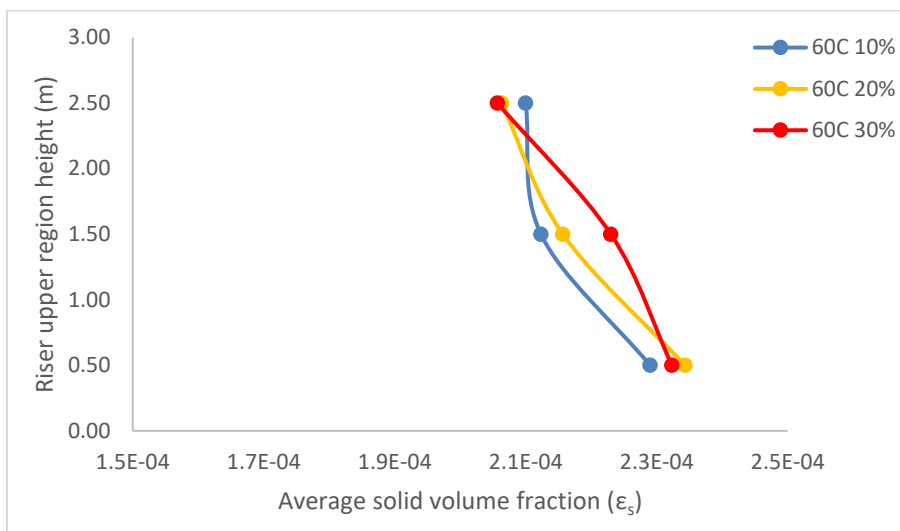


Figure 5-17: Ca(OH)_2 - Experimental average axial solids distribution under varying inlet gas conditions

From the axial distribution in Figure 5-17, it is observed that the average solids volume fraction at the bottom port is similar for all three gas relative humidity levels. While the case for 10% and 20%, follow the same trend, the average does increase slightly at the bottom and middle port when increase from 10% to 20%. For a relative humidity of 30%, it is observed that the average volume fraction only increases for the middle port while remaining very similar to that of 20% at the bottom and top port of the riser.

The axial solids distribution profile retrieved from the simulation is presented in Figure 5-18. The simulation enables the accessibility for solids volume fraction in the lower region of the riser as well. This way we can create an axial solids distribution curve within the riser.

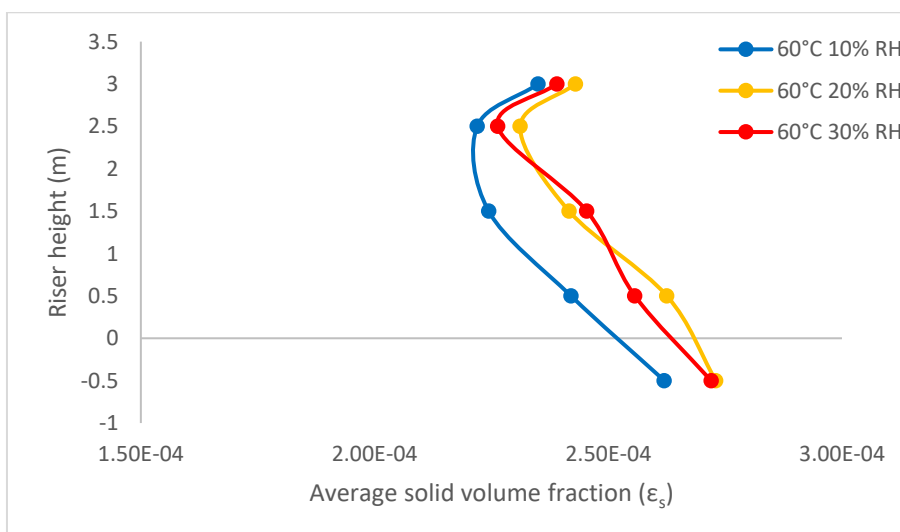


Figure 5-18: Ca(OH)_2 - Modelled average axial solids distribution under varying inlet gas conditions

The profiles at all of the different inlet relative humidity exhibit a slight C-shaped curve. It is seen that in the bottom region the concentration of solids decreases along the riser height until approximately at the top of the conical section. The increasing concentration of solids at the top of the riser was likely caused by the exit geometry and particles colliding with the top of the riser. This agrees with the findings from the radial distributions as well. From the axial distribution, it can be seen that the average solids volume fraction is very similar for all the conditions, especially in the upper region of the riser. The solids volume fraction also tends to be relatively constant in the upper region of the riser where the flow is more developed. It is also observed that for most of the riser height, a relative humidity of 30% has a lower average axial solids distribution than a relative humidity of 20%.

5.3.2 Potassium carbonate

As defined in the objectives, the influence of two independent variables on the hydrodynamics are investigated in this study for two sorbents. Therefore, the results pertaining to the experiments and simulations of potassium carbonate as sorbent will be discussed in this section. The discussion of potassium carbonate hydrodynamics and the influence and temperature and relative humidity will adapt the same structure as with the discussion of hydrated lime.

5.3.2.1 Influence of inlet gas temperature

(a) Pressure

In Table 5-12, the experimental results for the pressure differential while varying the inlet gas temperature are tabulated. The pressure differential across the riser (ΔP_{riser}) and the pressure differential across the upper region ($\Delta P_{\text{upper region}}$) are reported respectively.

Table 5-12: K_2CO_3 - Experimental pressure differential across riser and riser upper region under varying inlet gas conditions

Inlet gas relative humidity (%RH)	Inlet gas temperature ($^{\circ}\text{C}$)	ΔP_{riser} (Pa)	$P_{\text{upper region}}$ (Pa)
10	60	167.2	20.7
	70	153.4	18.5
	80	149.8	17.9
20	60	157.5	19.7
	70	152.1	16.4
	80	148.8	16.1

Inlet gas relative humidity (%RH)	Inlet gas temperature (°C)	ΔP_{riser} (Pa)	$P_{upper\ region}$ (Pa)
30	60	155.7	18.2
	70	151.5	15.4
	80	143.2	14.9

From the results reported above, it is observed that when the inlet gas relative humidity is kept constant while increasing the gas temperature from 60°C to 80°C, the pressure differential across both regions seem to decrease. As known from literature, pressure drop is largely influenced by the solids distribution in the riser. As a result, the decrease in pressure differential can mainly be ascribed to the solids distribution which is also influenced by the increase in temperature. In Section 5.3.2.1 (c), a more detailed discussion will be given on the solids distribution in the riser while varying the gas temperature. Furthermore, the gas temperature is known to influence gas density and gas viscosity. As the temperature increases, the gas density is decreased and the gas viscosity is increased. A warmer gas is known to be lighter, and thus exerts less pressure.

Comparing the pressure differential values of both regions, it is very prominent that the upper region pressure differential is only a fraction of the pressure differential across the riser. It can be expressed as approximately 12% of the total pressure losses. The large difference in values between these regions is the result of a venturi in the lower region of the riser.

For the simulations, a line probe was used to extract the pressure at different points in the riser. The main points of interest in the riser are the riser inlet, the conical area and the top of the riser. The modelled pressure values at each of the respective points as well as the pressure differential across the riser are reported in Table 5-13. For reference, the respective points are identified in Figure 5-9 earlier in Chapter 5.3.1.1, which also corresponds to Figure 3-6.

Table 5-13: K₂CO₃ - Modelled pressure at different points in the riser under varying inlet gas conditions

Inlet gas relative humidity (%RH)	Inlet gas temperature (°C)	(a): P_{inlet} (Pa)	(b): P_{cone} (Pa)	(c): P_{top} (Pa)	ΔP_{riser} (Pa)
10	60	-7.1	-549.7	-159.9	152.8
	70	-9.5	-553.1	-160.2	150.7
	80	-11.2	-557.8	-161.1	149.9
20	60	-8.7	-554.7	-162.3	152.6
	70	-10.3	-567.2	-161.6	151.3

	80	-12.7	-569.1	-159.3	146.6
	60	-10.5	-559.4	-161.3	150.8
30	70	-11.1	-569.2	-159.7	148.6
	80	-14.5	-573.1	-156.4	141.9

The modelled pressures at the inlet of the riser are relatively close to atmospheric pressure as the inlet is open and exposed to the environment. The pressure values from the conical area are significantly lower compared to the other regions. This would support the previously discussed finding that the majority of the pressure loss occurs in the lower region of the riser. The sudden drop in pressure can mainly be ascribed to the venturi, whereby the restriction causes intense flow acceleration and as a result the lower static pressure values. Another contribution would be the entry of solids particles at the exact same point as where the restriction is located, further contributing to the sudden drop in pressure as found in literature.

Considering the increase in gas inlet temperature, the decrease in pressure values are evident across the entire riser. It is also clearly realised that the pressure at the top of the riser is much lower than the pressure at the bottom of the riser, which is considered correct for vertical flow in the upward direction induced by a draught fan.

(b) Temperature

The optimal temperature profile would be to have a uniform temperature distribution in the reaction zone of the CFB. The experimental temperature in the riser was measured at four respective points along the riser, resembling the point identified on Figure 3-6. These values are reported in Table 5-14.

Table 5-14: K_2CO_3 - Experimental temperatures recorded at various points across the riser for varying inlet gas conditions

Inlet gas relative humidity (%RH)	Inlet gas temperature ($^{\circ}C$)	T_{inlet} (K)	T_1 (K)	T_2 (K)	T_3 (K)
10	60	334.9	313.3	334.0	333.4
	70	346.0	342.3	345.4	345.1
	80	356.0	345.4	355.1	353.0
20	60	334.0	312.6	333.1	332.6
	70	345.5	340.8	344.4	343.9
	80	355.3	344.1	353.8	352.2

30	60	333.6	312.8	332.8	331.8
	70	344.8	340.1	343.6	342.7
	80	354.9	342.8	353.6	351.4

It is seen that the inlet temperatures are all relatively close to the inlet gas temperature variable as this was specified according to the experimental plan. It is also observed that the temperatures at T_2 and T_3 , increases with approximately the same magnitude as the increase in inlet gas relative humidity, and it can be assumed that in a non-reacting CFB the inlet temperature will be one of the variables determining the temperature in the upper region of the riser. At T_1 , the temperature is lower compared to the rest of the riser. However, in the experimental results, the lowest temperature point in the riser could not be accessed, and as a result of the sudden drop in the conical area, the temperature at T_1 is starting to increase again and stabilising to form a uniform temperature distribution in the upper region. The temperature in the conical area will be addressed when looking at the modelled results. Furthermore, for all combinations of inlet gas temperature and relative gas humidity, a relatively uniform distribution exists in the upper region of the riser which is optimal for reacting flows.

The data presented in Table 5-15 was retrieved from the simulations of potassium carbonate with varying inlet gas temperatures. The respective points in the riser that were recorded to resemble the experimental results is identified on Figure 5-10 illustrated earlier in Chapter 5.3.1.1 for ease of tracking. The point also resembles reference Figure 3.6.

The modelled data shows clearly that there exists a direct relationship between the inlet gas temperature and the temperature in the riser, especially in the upper region. From this data, it was possible to record the temperature in the conical area of the riser.

Table 5-15: K_2CO_3 - Modelled temperatures recorded at various points across the riser for varying inlet gas conditions

Inlet gas relative humidity (%RH)	Inlet gas temperature ($^{\circ}C$)	T_{inlet} (K)	T_{cone} (K)	T_1 (K)	T_2 (K)	T_3 (K)
10	60	335.9	309.8	323.0	327.9	327.1
	70	346.0	320.2	338.1	342.8	340.9
	80	356.4	328.2	336.2	350.5	348.6
20	60	336.8	308.7	321.5	326.6	326.6
	70	345.9	320.0	337.4	342.7	340.1

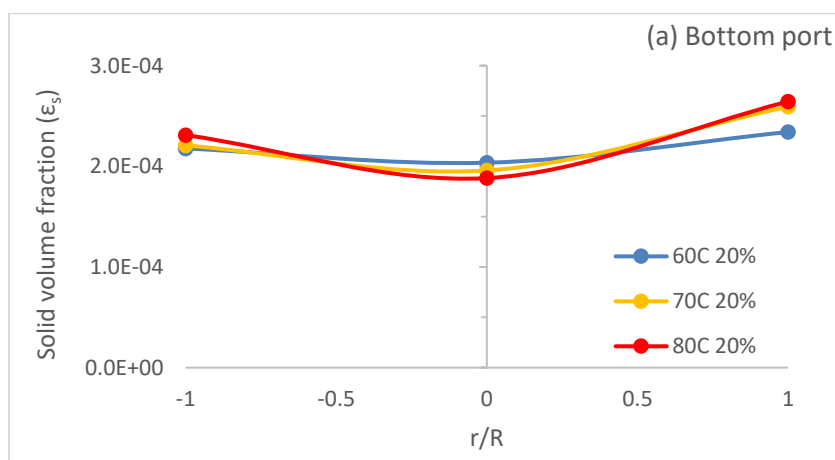
	80	356.2	327.9	345.4	350.3	347.8
30	60	334.6	307.1	319.7	324.9	325.1
	70	346.3	316.9	334.0	339.3	338.6
	80	356.7	325.0	334.8	346.9	346.4

The temperature recorded in the conical region are significantly lower and this could be ascribed to the solids entering the riser at the venturi throat. Heat transfer occurs from the warm gas to the colder solid particles and therefore the sudden temperature drop is seen. Temperature at point T_1 , T_2 and T_3 indicated that as the flow becomes more developed in the upper region of the riser, the temperature tends to stabilise at a temperature that is within close approximation of the inlet gas temperature. As a result, a uniform temperature distribution exists in the upper region of the riser.

(c) Solids distribution

The solids distribution of potassium carbonate while varying the inlet gas temperature will be discussed considering two distributions. These are the radial distribution at the three (3) sampling ports and the average axial solids distribution in the upper region of the riser. Three radial positions across the diameter of the riser, at three axial positions in the upper region were used to calculate the local solids volume fraction. From this data the radial and average axial solids distribution is illustrated.

In this section, the results pertaining the variation of inlet temperature and an inlet gas relative humidity of 20% will be discussed. The remainder of the results can be viewed in Annexure E, as the trends and findings are similar to what will be discussed here.



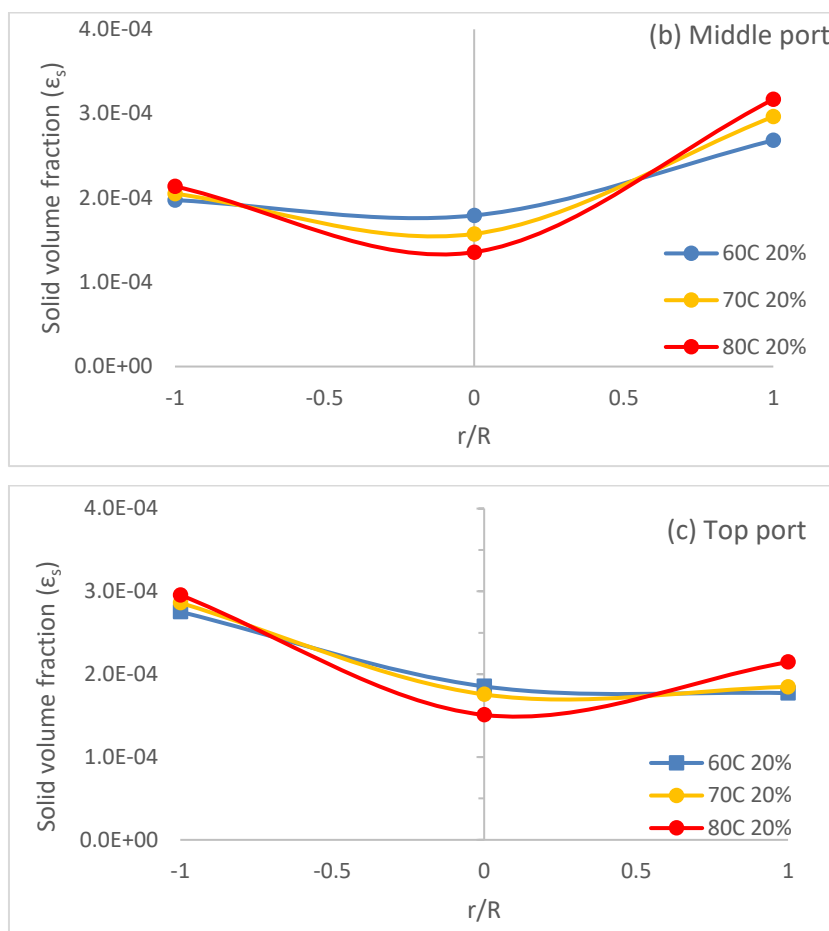


Figure 5-19: K_2CO_3 – Experimental radial solids distribution profile at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet conditions

In Figure 5-19, the radial solids volume fraction profile is illustrated at (a) bottom port, (b) middle port and the (c) Top port. The graphs represent the solids volume fraction on the y-axis and the radial position on the x-axis with 0 indicating the centre of the riser.

The radial distribution in the riser will be discussed with reference to Figure 3-3 and Figure 3-4. Considering the angle at which the solids enter with regard to the vertical axis, the momentum of the solids particles together with the gas flow, result in the particle flow being transported along the opposite wall from the solids inlet for the largest part of the riser. This direction of particle flow will explain the observation of a slight asymmetrical distribution to the right at the bottom and middle port. The opposite distribution profile is seen at the top port, which can be as a result of the T-shaped outlet at the top of the riser. Although the slight asymmetrical distribution is definitely observed, the difference in solid concentrations at the respective radial positions is significantly small, indicating a rather flat radial distribution, especially at the bottom port.

The influence of varying inlet gas temperature is observed, in the radial distribution. At all three ports, it is seen that increasing the inlet gas temperature results in the solids volume fraction in

the core being decreased and the solids volume fraction being increased near the wall of the riser. This agrees well with findings from literature. It is, however, still not extremely apparent, but definitely more compared to previous results pertaining to hydrated lime. The low solids flux does influence the ability to see the effect clearly. An unfortunate observation is the inability to capture the typical core-annulus from the experimental data which can be ascribed to the low solids flux used to carry out experiments and simulations.

In Figure 5-20, the radial distribution profiles from the CFD simulations are presented.

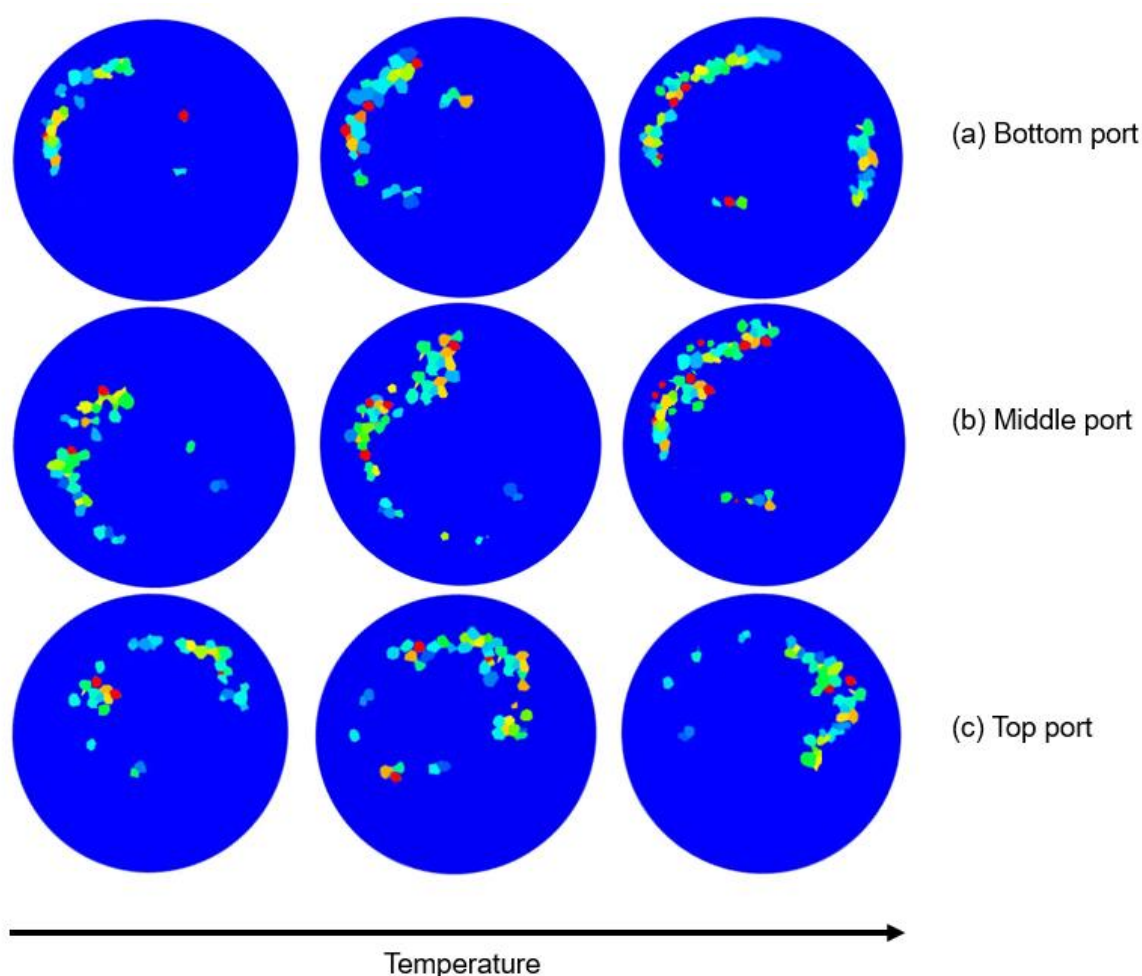


Figure 5-20: K_2CO_3 – Modelled radial solids volume fractions at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas conditions

The observations from experimental results can be confirmed and can visually be seen. It is seen that when the inlet gas temperature increases (left to right) the area covered by solid parcels in the core of the riser is decreased, and the area covered near the wall is increased. The migration of flow can also be seen when comparing the top port to the other two ports.

To further describe the solids distribution under varying inlet gas temperature conditions, the axial solids distribution is illustrated in Figure 5-21. The axial distribution is given only for the upper

region of the riser, as this was the only part being accessible during operation. The axial distribution is given in the same format as being discussed earlier with the results for hydrated lime.

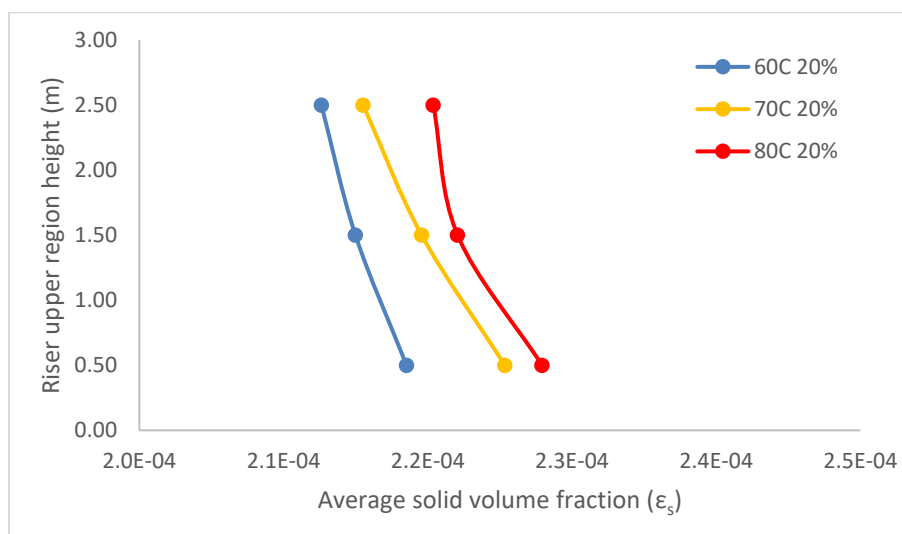


Figure 5-21: K₂CO₃ - Experimental average axial solids distribution for varying inlet gas conditions

It is observed that the axial solids distribution in the upper region of the riser exhibits a uniform distribution between the bottom port and the upper port. The difference in average solids volume fraction between the top and bottom of the riser is approximately 0.7×10^{-5} . The observations agree well with the findings in literature that the average solids volume fraction increases with an increase in temperature.

The average axial solids volume fraction profiles from the CFD simulations was extracted by creating circular planes in STAR-CCM+[®] which reports the surface average solids volume fraction across the planes created. The reported average axial volume fraction profiles are illustrated in Figure 5-22.

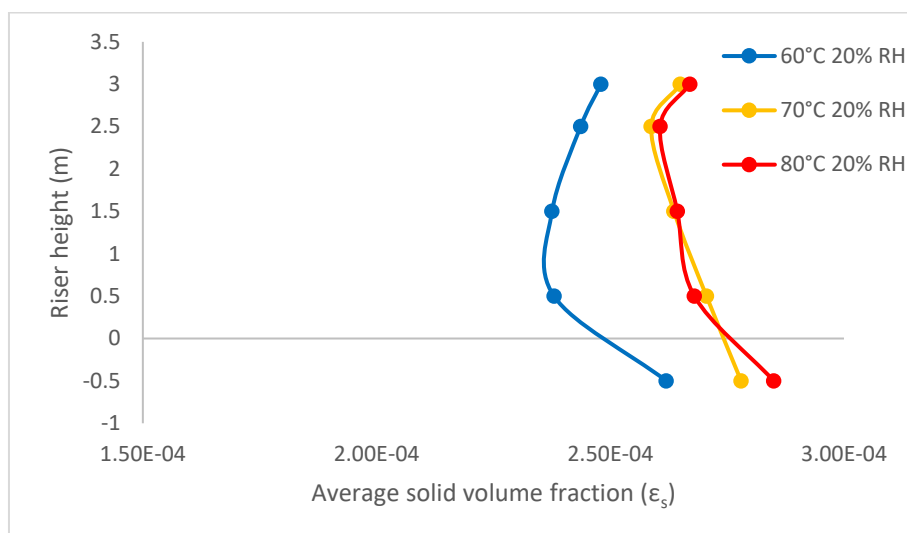


Figure 5-22: K_2CO_3 - Modelled average axial solids distribution for varying inlet gas conditions

The average axial profiles exhibit a strong C-shaped curve along the height of the riser. This type of distribution is commonly reported in literature, however, it is thought that especially in the bottom of the riser the distribution is a result of the increasing diameter of the conical area, rather than flow development. The increase seen at the top of the riser is a result of the T-shaped outlet, and particles colliding with the top of the riser.

5.3.2.2 Influence of inlet gas relative humidity

(a) Pressure

The influence of varying the inlet gas relative humidity was investigated by considering the pressure differential across two regions of the riser. These regions were clearly identified during previous discussions and is illustrated in Figure 3-6. The experimental values are reported in Table 5-16.

Table 5-16: K_2CO_3 - Experimental pressure differential across riser and riser upper region for varying inlet gas conditions

Inlet gas temperature (°C)	Inlet gas relative humidity (%RH)	ΔP_{riser} (Pa)	$\Delta P_{upper\ region}$ (Pa)
60	10	167.2	20.7
	20	157.5	19.7
	30	155.7	18.2
70	10	153.4	18.5
	20	152.1	16.4

Inlet gas temperature (°C)	Inlet gas relative humidity (%RH)	ΔP_{riser} (Pa)	$\Delta P_{upper\ region}$ (Pa)
80	30	151.5	15.4
	10	149.8	17.9
	20	148.8	16.1
	30	143.2	14.9

From the tabulated values, it is clearly seen that the pressure differential decreases as the inlet gas relative humidity increases, for both of the regions. The changes in pressure can be ascribed to various reasons, one being the change in solids distributions and a second reason being the water vapour content of the gas being increased as the relative humidity is increased. The increasing water vapour content results in the density of the gas being decreased and consequently, the pressure is decreased. As with all other results, it can immediately be observed that the majority of pressure losses occur within the lower region of the riser as the upper region pressure differential in this case can be expressed as approximately 12%.

The modelled pressure values as extracted from the CFD simulations are reported in Table 5-17.

Table 5-17: K_2CO_3 - Modelled static gauge pressure and modelled pressure differential across the riser

Inlet gas temperature (°C)	Inlet gas relative humidity (%RH)	(a): P_{inlet} (Pa)	(b): P_{cone} (Pa)	(c): P_{top} (Pa)	ΔP_{riser} (Pa)
60	10	-7.1	-549.7	-159.9	152.8
	20	-8.7	-554.7	-162.3	152.6
	30	-10.5	-559.4	-161.3	150.8
70	10	-9.5	-553.1	-160.2	150.7
	20	-10.3	-567.2	-161.6	151.3
	30	-11.1	-569.2	-159.7	148.6
80	10	-11.2	-557.8	-161.1	149.9
	20	-12.7	-569.1	-159.3	146.6
	30	-14.5	-573.1	-156.4	141.9

Similar to all previously discussed pressure results, the pressure at the inlet is close to atmospheric pressure which is correct in the sense that the inlet is exposed to ambient conditions. The sudden pressure drop in the conical area is already known to exist as a result from the venturi

in the lower region of the riser. The reported values exhibit the same trend of reflecting the influence of varying inlet gas relative humidity as pressure decreases as relative humidity increases. The modelled pressure and the pressure differentials indicates the similar trends and findings as with experimental observations.

(b) Temperature

Temperature and gas relative humidity are inversely proportional to each other. It is also evident in this study when analysing the results. The experimental results for varying inlet gas relative humidity are tabulated in Table 5-18. The respective point as identified on Figure 3-6 have been reported.

Table 5-18: K_2CO_3 - Experimental temperatures at different points across the riser for varying inlet gas conditions

Inlet gas temperature (°C)	Inlet gas relative humidity (%RH)	T_{inlet} (K)	T_1 (K)	T_2 (K)	T_3 (K)
60	10	334.7	308.5	330.8	329.9
	20	333.8	307.9	329.9	329.1
	30	333.4	308.1	329.6	328.3
70	10	345.8	337.1	342.1	341.5
	20	345.3	335.6	341.1	340.3
	30	344.6	334.9	340.3	339.1
80	10	355.8	340.2	351.7	349.3
	20	355.1	338.9	350.4	348.5
	30	354.7	337.6	350.2	347.7

From the values reported in Table 5-18, the decrease in temperature in the upper region of the riser is clearly seen as a result of increasing gas relative humidity. The indirect relationship exists due to the water vapour content of the gas that increases which causes the gas temperature to decrease.

The modelled temperatures that resemble the respective temperature points as reported with the experimental results, are presented in Table 5-19.

Table 5-19: K_2CO_3 - Modelled temperatures at different points across the riser for varying inlet gas conditions

Inlet gas temperature ($^{\circ}C$)	Inlet gas relative humidity (%RH)	T_{inlet} (K)	T_{cone} (K)	T_1 (K)	T_2 (K)	T_3 (K)
60	10	335.9	309.8	323.0	327.9	327.1
	20	336.8	308.7	321.5	326.6	326.6
	30	334.6	307.1	319.7	324.9	325.1
70	10	346.0	320.2	338.1	342.8	340.9
	20	345.9	320.0	337.4	342.7	340.1
	30	346.3	316.9	334.0	339.3	338.6
80	10	356.4	328.2	336.2	350.5	348.6
	20	356.2	327.9	345.4	350.3	347.8
	30	356.7	325.0	334.8	346.9	346.4

Generally, the temperature exhibits a uniform temperature distribution in the upper region which can be seen by both the experimental and modelled temperature values in Table 5-18 and Table 5-19 respectively. It is also observed that the decrease in temperature is significantly small for the increase in gas relative humidity. It can be due to the range of relative humidity in combination with the temperatures used in this study.

(c) Solids distribution

The influence of varying the inlet gas relative humidity is studied by comparing two solid distribution profiles, one in the radial direction and one in the axial direction. For the radial solids distribution, the local solids volume fraction calculations are used. These were calculated using the sample weights that were collected during the operation. Again, only the data for operating conditions of varying inlet gas relative humidity and 70 $^{\circ}C$ are going to be discussed. The additional data can be found in Annexure E. In Figure 5-23, the radial distribution of solids volume fractions is given at the three respective sampling ports being the bottom port (a), the middle port (b) and the top port (c).

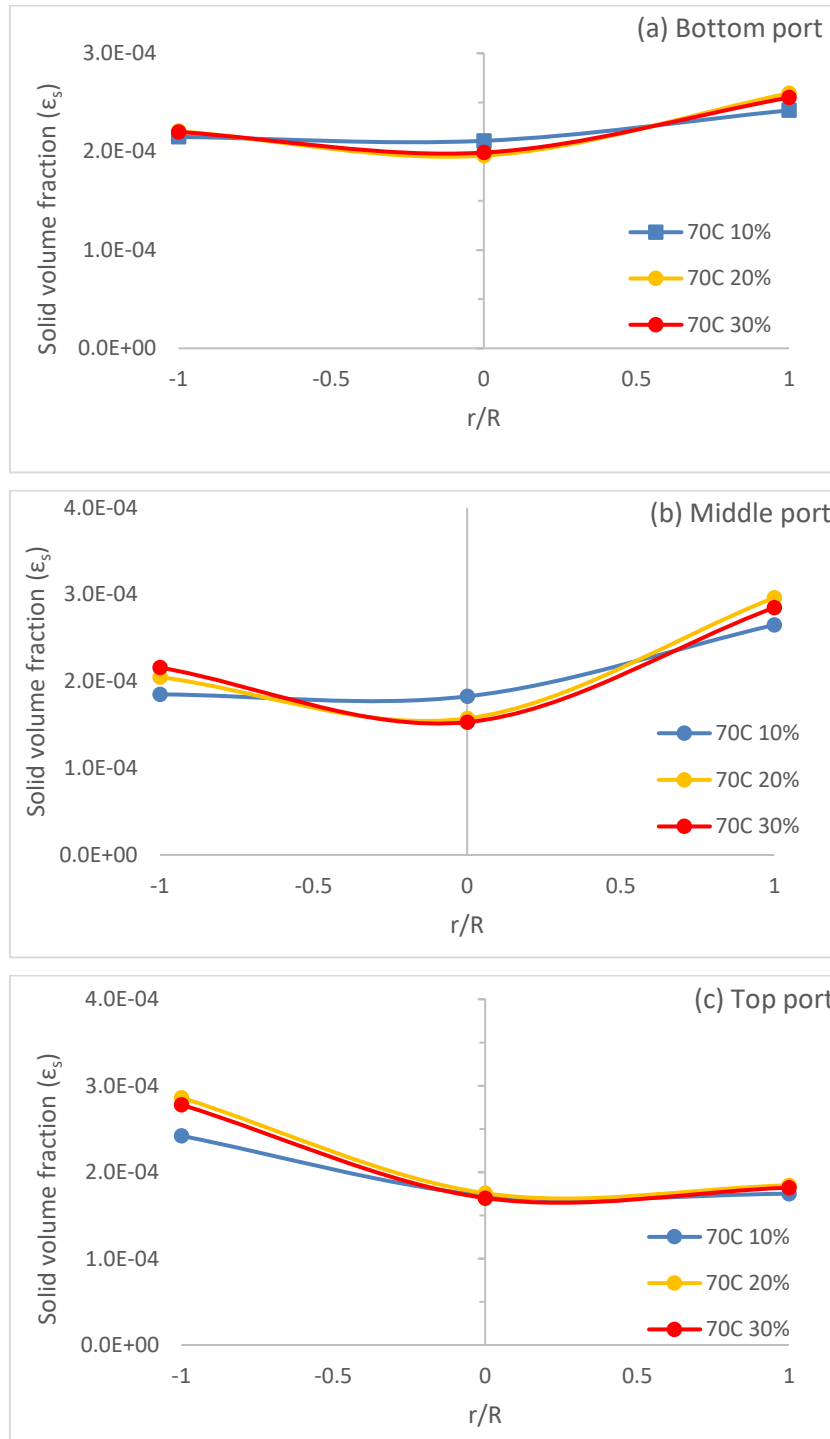


Figure 5-23: K_2CO_3 - Radial solids distribution profile at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas conditions

The solids volume fraction is presented on the y-axis, while the radial position is presented on the x-axis. Each data point resembles the solids volume fraction at a specific radial position. The different coloured data point represents the different operating conditions. In Chapter 3.2.1, Figure 3-3 gives an illustration indicating the different inlets and the outlet of the riser with respect to each other. This illustration will be used as reference for the discussion of the radial solids distribution.

The influence of gas relative humidity is still a developing topic within the research field and its direct influence is not well known from literature. However, the influence of gas relative humidity is often linked with the particle-particle and particle-wall electrostatic forces in two-phase flow. From the data it is seen that with an increase in gas relative humidity from 10% to 20% at a constant inlet temperature of 70°C, the concentration of solids in the core region seems to decrease slightly. When the gas relative humidity is further increased to 30%, the opposite is seen to where the concentration of solids in the core seen to increase slightly. This agrees with a source from literature, in which different humidity levels were evaluated. The increase in gas relative humidity causes the particle-particle and particle-wall electrostatic forces to decrease.

Visually, the radial solids distribution profiles as retrieved from the CFD simulations illustrate the findings very clear. These are illustrated in Figure 5-24. The area covered near the wall of the riser increase for an inlet gas relative humidity increase from 10% to 20%. However, the area covered near the wall slightly decreases as the inlet gas relative humidity is further increased to 30%. These support the findings from experimental data.

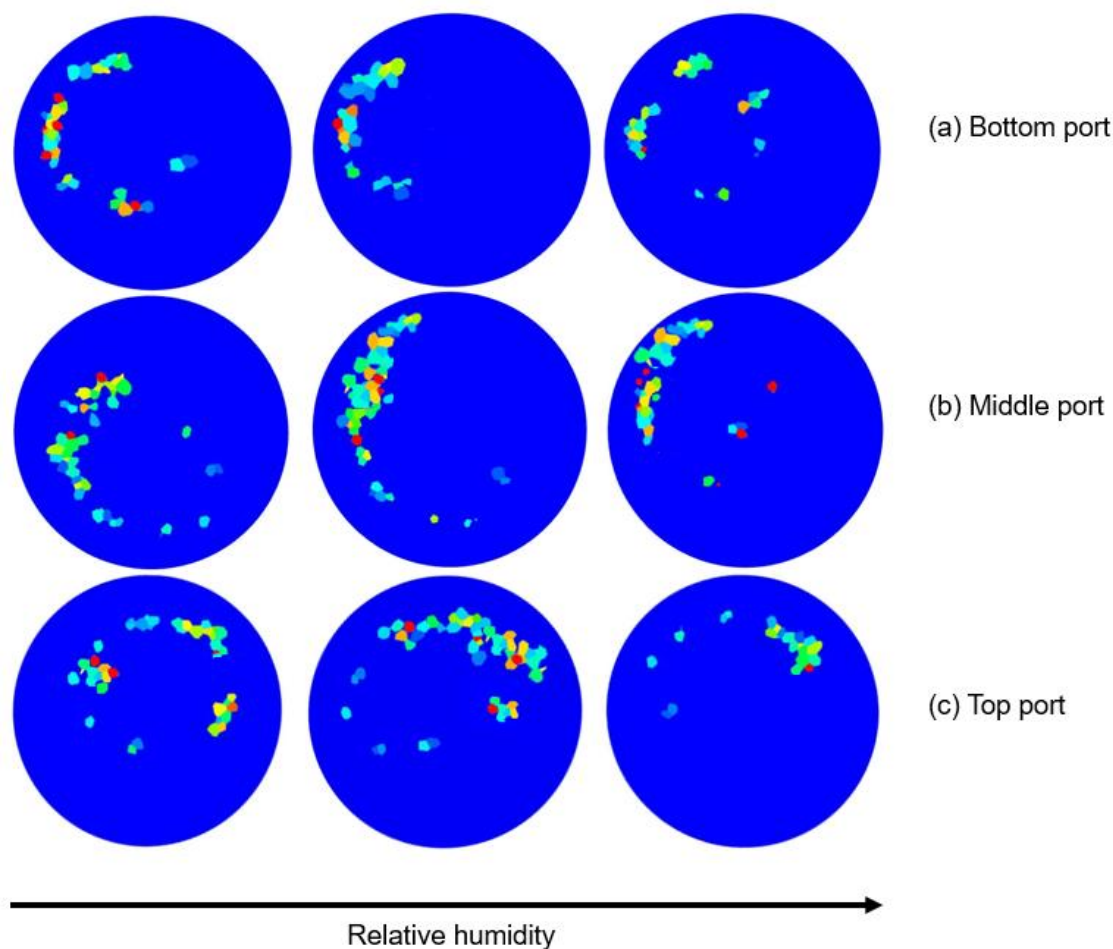


Figure 5-24: K_2CO_3 – Modelled radial solids volume fractions at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas conditions

Figure 5-25 displays the experimental average axial solids volume fraction distribution profile for varying inlet gas relative humidity at an inlet gas temperature of 70°C, While Figure 5-26 presents the modelled data for the same conditions.

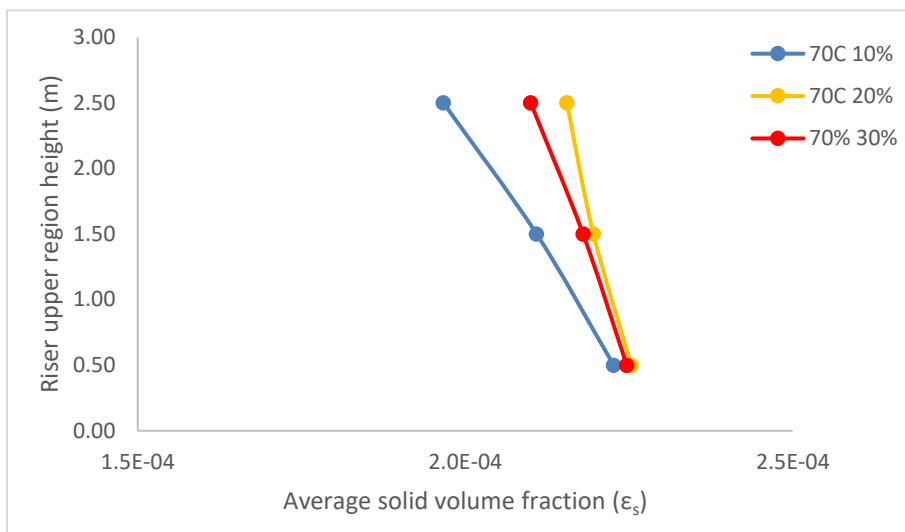


Figure 5-25: K₂CO₃ - Experimental average axial solids distribution for varying inlet gas conditions

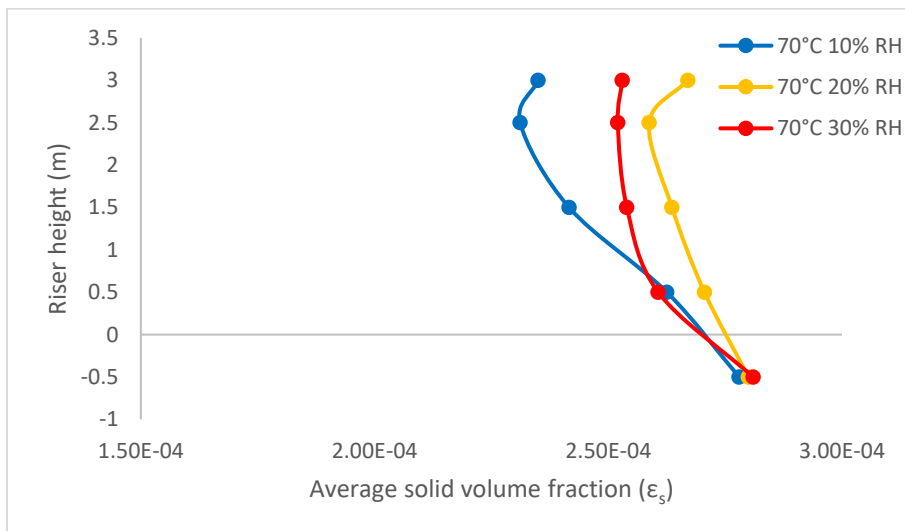


Figure 5-26: K₂CO₃ - Modelled average axial solids distribution for varying inlet gas conditions

In Figure 5-25 and Figure 5-26, the y-axis represents the riser height from height = 0m which is the reference point as indicated in Chapter 3.4.4 and the x-axis represents the average solids volume fraction. It is seen that especially for the top port, there does exist some difference in the average axial solids volume fractions for different levels of inlet gas relative humidity. It is also significant to see the same trend as with the radial distribution, that the average solids volume fraction increases when the inlet gas relative humidity increases from 10% to 20% but decreases when the relative humidity is further increased. For the bottom port, it is not as prominent as with the top port of the riser. It is seen from the modelled average solids volume fractions in Figure 5-

26, it exhibits a slight C-shaped curve when the entire riser is considered. This is a common axial solids distribution profile found in CFBs from literature (Panday *et al.*, 2016).

5.4 Comparison of results for hydrated lime and potassium carbonate

Seeing that the sorbent properties as determined in Section 5.1 is quite different for the two respective sorbents, a slight different behaviour in the CFB is expected. For this purpose, data retrieved for hydrated lime and potassium carbonate have been compared to analyse if a difference could be captured. For this section, results pertaining to the pressure differential across the riser, as well as the upper region, the temperature at different points and the radial solids distribution have been considered. In Figure 5-27, the pressure differential across the riser, as well as the pressure differential across the upper region has been illustrated for both sorbents for all the different experiments. Hydrated lime (HL) is indicated with a solid line and potassium carbonate (PC) with a dashed line. It is observed that the pressure differential across the riser when potassium carbonate is used as sorbent is higher in all of the experiments. The calculated pressure drops are also more consistent for all of the experiments with potassium carbonate compared to that of hydrated lime.

For the pressure differential across the upper region, the reported values for both the sorbents are rather consistent for all the experiments and relatively close in value. The pressure differential in this area for all experiments also differs with approximately 8 Pa. It is realised that the type of particles have a bigger influence on the pressure differential in the lower region of the riser compared to the upper region.

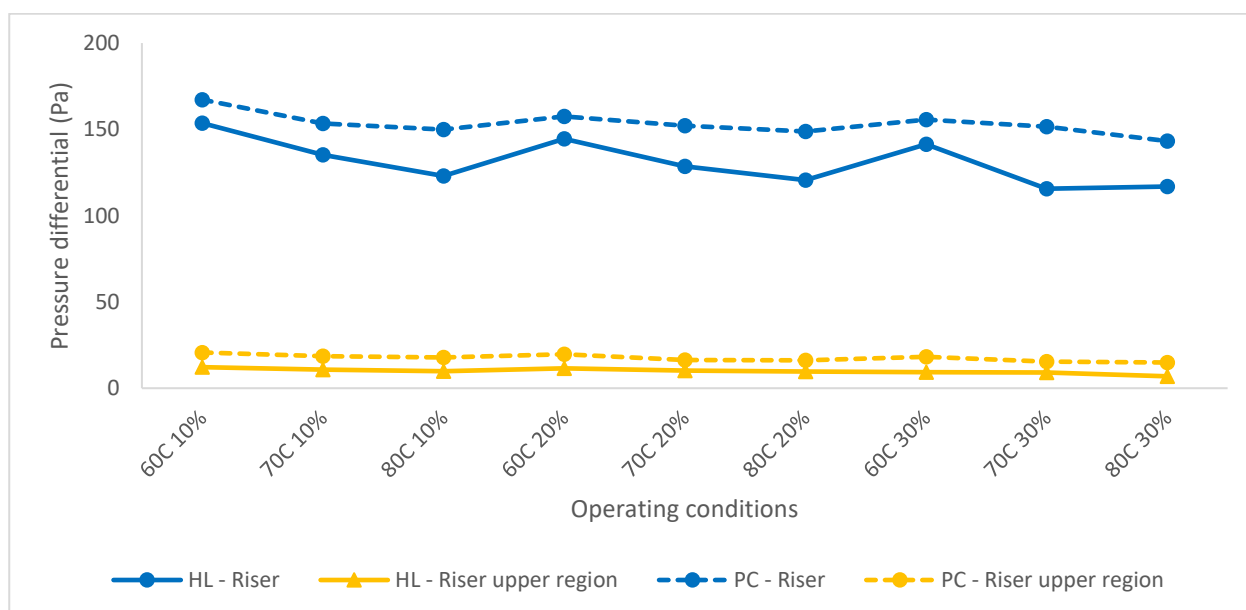


Figure 5-27: Pressure differential comparison of hydrated lime (HL) and potassium carbonate (PC) across the riser and riser upper region

When comparing the temperature in the riser for the respective sorbents, only the experimental results pertaining to the conditions of varying inlet gas temperature at a fixed inlet gas relative humidity of 20% are reported in Figure 5-28.

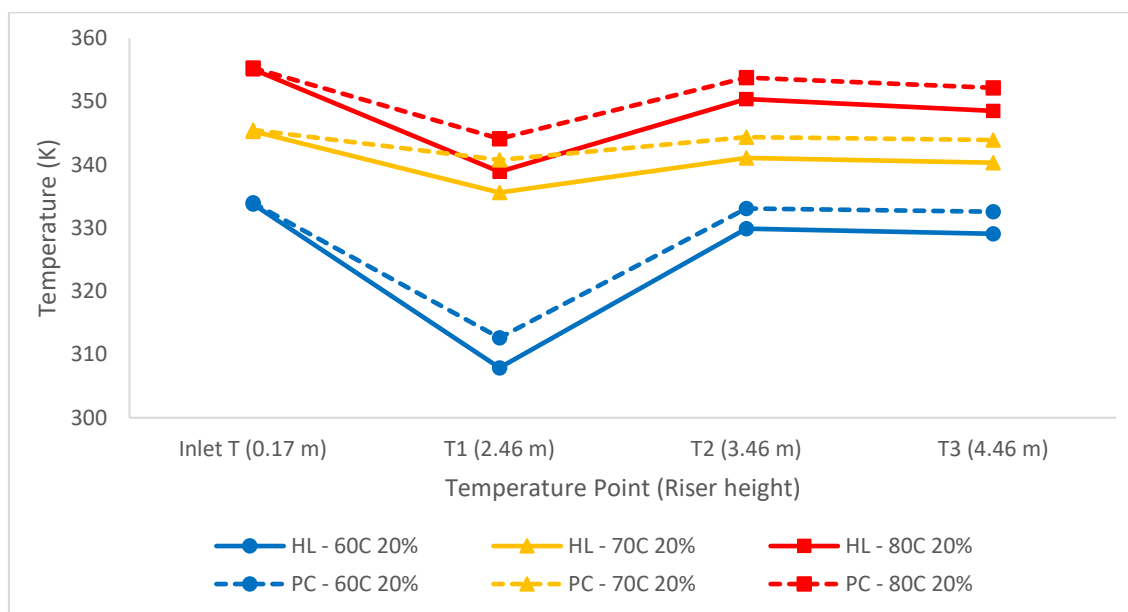


Figure 5-28: Temperature comparison of hydrated lime (HL) and potassium carbonate (PC)

It is clearly observed that the temperature in the riser is slightly higher when potassium carbonate is used as sorbent compared to the results of hydrated lime. There exists a relatively consistent deviation in values for the respective sorbents throughout the riser.

Lastly, the radial solids distribution of both sorbents has been compared at all three of the riser ports. For comparison purposes, only the results pertaining to the operating conditions of a fixed inlet gas relative humidity of 20% and varying inlet gas temperature has been illustrated in Figures 5-29, 5-30 and 5-31. In Figure 5-32 the radial solids distribution at the bottom port is illustrated. It is seen for all operating conditions that deviation between the solids volume fraction in the core zone ($r/R = 0$) are the smallest and is consistent in magnitude as the inlet gas temperature increases. At the radial position $r/R = -1$, the deviation is the biggest for 70°C and 20% RH. Furthermore, the deviation at $r/R = 1$ decreases as the temperature increases. For all the conditions it is also observed that the solids volume fraction for potassium carbonate is higher in the core zone of the riser, but lower near the wall of the riser.

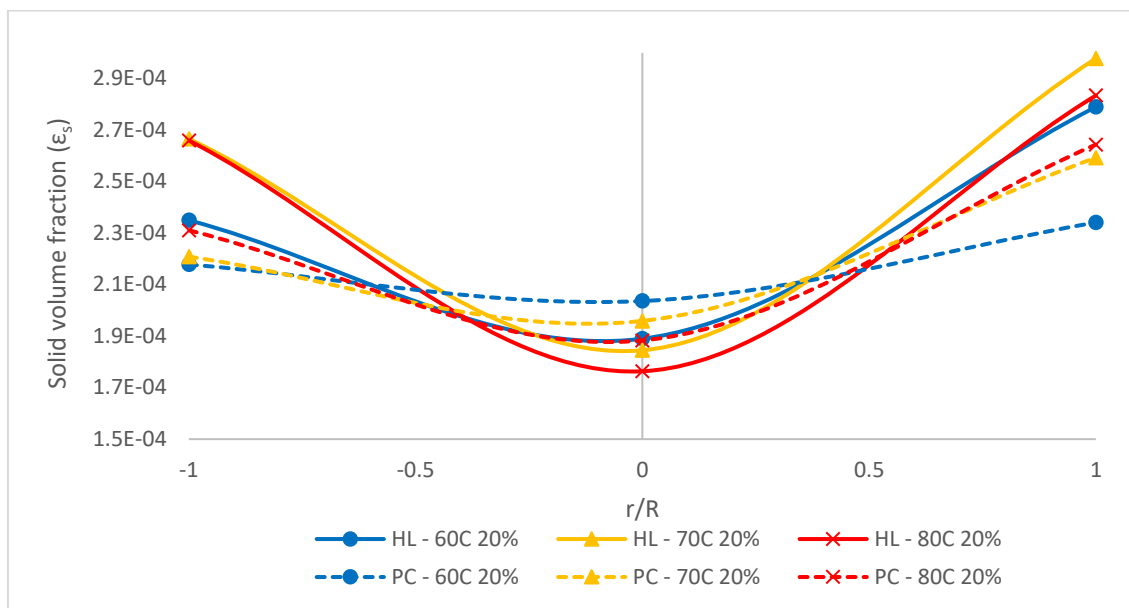


Figure 5-29: Radial solids distribution comparison of hydrated lime (HL) and potassium carbonate (PC) at the bottom port of the riser

In Figure 5-30, the radial solids distribution at the middle port of the riser is illustrated for both of the sorbents. Again, the solids volume fraction for hydrated lime is higher near the wall area of the riser and lower in the core zone of the riser compared to that of potassium carbonate. The deviations for the conditions of 80°C and 20% RH are the smallest in all radial positions. From the figure, it is observed that potassium carbonate has a more homogeneous distribution than hydrated lime.

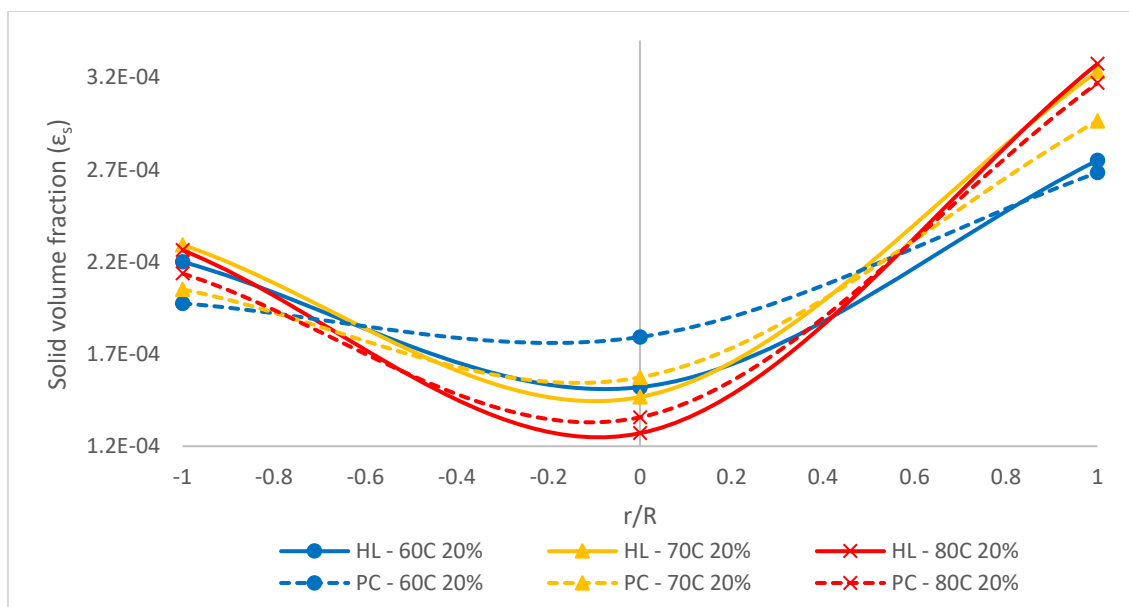


Figure 5-30: Radial solids distribution comparison of hydrated lime (HL) and potassium carbonate (PC) at the middle port of the riser

Lastly, the radial solids distribution of both sorbents at the top port of the riser are illustrated in Figure 5-31. It is observed that the solids volume fraction for hydrated lime is higher compared to

that of potassium carbonate in all radial position, except for the conditions of 60°C and 20% RH at $r/R = -1$. Furthermore, for 70°C and 20% RH, the deviation between the two sorbents are higher near the wall area than in the core zone of the riser. The complete opposite trend is seen for 60°C and 20% RH, where the deviation is the biggest in the core zone compared to extremely small deviation near the wall.

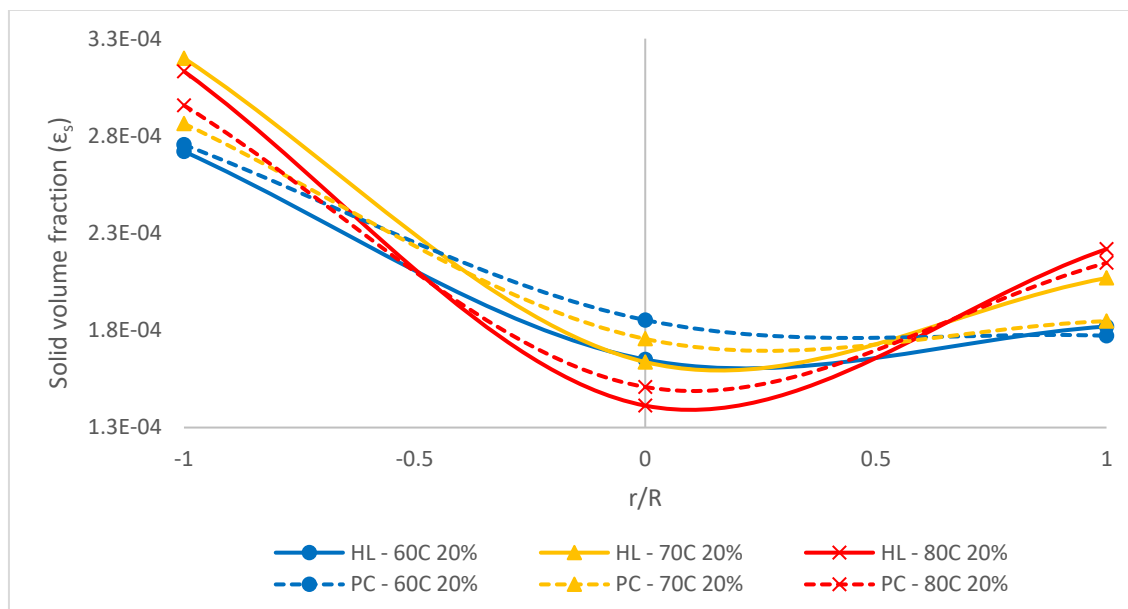


Figure 5-31: Radial solids distribution comparison of hydrated lime (HL) and potassium carbonate (PC) at the top port of the riser

From the comparisons made above, there does exist some clear differences between the data for hydrated lime and potassium carbonate, although not to the extent expected. It can ultimately be assumed that the type of particle will influence the hydrodynamics in a CFB riser.

CHAPTER 6 CONCLUSION AND RECOMMENDATIONS

This chapter summarises the findings and main conclusions of the study on the effect of humidity and temperature on the hydrodynamics of different sorbents. A list of recommendations is also included to address current issues and improve experimental results for future work.

6.1 Conclusion

The first objective of this study was formulated to determine the particles properties of hydrated lime ($\text{Ca}(\text{OH})_2$) and potassium carbonate (K_2CO_3) that were significant for the CFD simulations. The conclusions that were made from this are as follows:

- The two sorbents have significantly different properties in terms of particle size and densities.
- Hydrated lime can be classified according as a Geldart Group C powder which is the most difficult group to fluidise and tend to behave more like clusters.
- Potassium carbonate can be classified as a Geldart Group A powder, which is the most preferred choice for fluidisation applications due to their ability to easily be fluidised.

The second objective of this study was to develop a representative and validated model that could be employed in describing the hydrodynamics of both hydrated lime, and potassium carbonate. The conclusions drawn related to this objective are as follows:

- The verified model as discussed in Section 4.1.1 was deemed sufficient in predicting the hydrodynamics based on theoretical means.
- The modelled data was validated against the experimental data with the same operating conditions for both hydrated lime and potassium carbonate, and showed satisfactory correlation between the data for all variables. The model was therefore assumed to be sufficient for describing the hydrodynamics.
- Overall, for both the pressure differential across the riser and temperature in the riser of hydrated lime, the modelled values were slightly lower than that of the experimental results.
- The pressure differential across the riser reported for hydrated lime, showed an average deviation of 8.2% between the modelled data and the experimental data corresponding to approximately 10 Pa.
- The modelled pressure differential across the riser for potassium carbonate was slightly lower compared to the experimental results. For three conditions: 80°C 10% RH, 70°C 20% RH and 80°C 30% RH, the modelled and experimental values had a significantly high agreement with deviations calculated to be -0.1%, 0.5% and 0.9% respectively. The average deviation of the modelled data from the experimental data was calculated to be 2%.

- The temperature data sets agreed well, except for the temperature at T_1 , which showed the largest deviations. The deviations at T_1 ranges between -3.9% and 2.1%. From the R^2 values, it is clearly observed that the agreement of experimental data and modelled data is satisfactory with most values exceeding a value of 0.9.
- The temperature recorded for potassium carbonate exhibits an overall good agreement. The same observation as with hydrated lime was made for temperature at point T_1 , showing the biggest deviations. The largest deviation was calculated to be -4.7% for the run of 60°C and 10% RH. For most operating conditions the R^2 values exceeds 0.8 which indicate a good correlation in data.
- The modelled radial solids distributions for hydrated lime at the bottom port and top port of the riser showed satisfactory agreement with experimental data. The R^2 -values for the different radial positions were determined to be 0.9295, 0.6683 and 0.9597 for the bottom port, and 0.8883, 0.1529 and 0.947 for the top port respectively. It is observed that the core zone at the top port showed poor correlation which can be ascribed to the chaotic flow in the top of the riser due to the T-shaped outlet.
- The modelled radial solids distributions at both the bottom and top port of the riser showed reasonable agreement with experimental data. The R^2 -values for the different radial positions were determined to be 0.7241, 0.6512 and 0.6081 for the bottom port, and 0.7487, 0.5702 and 0.437 for the top port respectively.

The third and main objective of this study was to assess the influence of the inlet gas temperature and the inlet gas relative humidity on the hydrodynamics of hydrated lime and potassium carbonate. The conclusions drawn related to this objective are as follows:

For hydrated lime, the influence of varying the inlet gas temperature and relative humidity are:

- The pressure differential across the riser as well as the pressure differential across the upper region of the riser decreases slightly for an increase in gas temperature as well as an increase in relative humidity.
- The temperature in the riser exhibits a uniform distribution along the length of the riser for varying both the inlet gas temperature and relative humidity. The temperature in the top of the riser was in close approximation with the inlet gas temperature for all operating conditions. An increase in inlet gas relative humidity, exhibits a significantly small decrease in riser temperature, as a result of the water vapour fraction of the air being increased.
- An increase in gas temperature results in a slight decrease in solids concentration in the core zone of the riser. Consequently, an increase in solids concentration is seen near the riser wall.
- Solids concentration in the core zone of the riser decreased for an inlet gas relative humidity increase from 10% to 20%. Further increasing the inlet gas relative humidity from 20% to 30%,

resulted in the solids concentration in the core zone to increase slightly. The opposite trend is seen in the wall area of the riser. This is as a result of the particle-particle and particle-wall electrostatic forces that decreases with an increase in gas relative humidity.

- Axially, the solids concentration in the upper region of the riser is relatively constant throughout. A slight increase in the average solids volume fraction is observed at all the riser ports for an increase in inlet gas temperature.
- The axial solids distribution showed different trends for each of the riser ports with an increase in relative humidity. At the bottom port, the average solids volume fraction increases with inlet gas relative humidity increasing from 10% to 20%. A further increase from 20% to 30% resulted in a decrease in average solids volume fraction. At the middle port, an increase is seen throughout for increasing inlet gas relative humidity. Lastly, the top port showed a decrease throughout for an increase in inlet gas relative humidity.
- The typical core-annulus flow could not be identified for any of the operating conditions and could be a result of the combination of operating conditions used in the study.

The varying inlet gas temperature and relative humidity have the following influences on the hydrodynamics of potassium carbonate:

- An increase in any of the two variables results in a decrease in the pressure differential across the riser, as well as the pressure differential across the upper region of the riser. It can mainly be ascribed to changes in gas density and viscosity, as well as changes in solids distribution when the variables are increased.
- The temperature in the riser exhibits a uniform distribution, with temperature in the top of the riser being relatively close to the inlet temperature.
- The temperature in the riser and the inlet gas relative humidity are inversely proportional to each other. An increase in the inlet gas relative humidity results in the temperature throughout the riser to slightly decrease. The reason is as the relative humidity increases, the water vapour content of the gas increases resulting in a temperature decrease.
- The radial solids distribution exhibits a decrease in solids concentration in the core zone of the riser and an increase in solids concentration near the wall of the riser with an increase in the inlet gas temperature.
- The radial solids distribution for varying gas relative humidity exhibits the same trend as with hydrated lime. The solids concentration in the core zone of the riser decreases with increasing inlet gas relative humidity from 10% to 20%, while a further increase from 20% to 30% results in a decrease in solids concentration in the core zone. This can be ascribed to the influence of relative humidity on the particle-particle and particle-wall electrostatic forces.

- For the axial distribution, an increase in the average solids volume fraction is observed for all three riser ports when the inlet gas temperature is increased.
- The axial solids distribution shows the average solids volume fraction at the bottom of the riser are very similar for all the levels of inlet gas relative humidity, while the difference increases at the other ports. It is still observed that the higher the inlet gas relative humidity, the higher the average solids volume fraction.

Lastly, a swift comparison between the two sorbents, hydrated lime and potassium carbonate has been made. The following conclusions have been drawn from this:

- The pressure differential across the riser as well as the pressure differential across the upper region of the riser are slightly higher for potassium carbonate compared to hydrated lime.
- The temperature in the riser is slightly higher for potassium carbonate, than for hydrated lime. The difference is however very small.
- For the radial distribution, the solids volume fraction near the wall of the riser is higher for hydrated lime, than for potassium carbonate, while the opposite is seen in the centre of the riser where the solids volume fraction for potassium carbonate is higher than that of hydrated lime.

The above conclusions, indicate that the particle properties have an influence on the hydrodynamics. However, the influence was expected to be more evident than what was seen by the results.

6.2 Recommendations

To improve the results and outcomes of future studies related to this topic the following recommendations are made:

Although small, the deviations in the modelled data from the experimental data does leave room for improvements in future studies. It is especially essential to the pressure differentials and solids distribution in the riser that other drag models are explored to predict the drag force. The employment of a heterogeneous drag model could lead to an overall better approximation of the drag force and improve the results.

Recommendations pertaining to the experimental work are made to improve the facility and to simplify operation and control of the different variables. Although the process conditions could be met, it required a lot of calculations based on ambient conditions which vary each day. Additionally, the manual control of the inlet gas temperature and the inlet gas relative humidity with manual measurements are time consuming and leave a lot of room for human error. The installation of additional thermocouples and moisture sensors with digital output can improve

future work tremendously. It can also enable the researcher to access real-time data, especially for the humidity inside the riser which could not be recorded during this study. It should also be considered exploring other methods to humidify the inlet gas, as the current method has a very small operating range.

Furthermore, the average solid volume fraction can be improved significantly if more radial solids samples are taken across the entire cross section of the riser.

BIBLIOGRAPHY

Abbasi, E. & Arastoopour, H. 2013. CFD Simulation of CO₂ Sorption in a Circulating Fluidized Bed Using Deactivation Kinetic Model. , in: Eds., P. (Ed.), 10th International Conference on Circulating Fluidized Beds and Fluidization Technology. ECI Digital archives. . ECI Digital archives.

Adamczyk, W.P., Klimanek, A., Białecki, R.A., & Gabriel, W. 2014. Particuology Comparison of the standard Euler – Euler and hybrid Euler – Lagrange approaches for modeling particle transport in a pilot-scale circulating fluidized bed. *Particuology*. 15:129–137.

Almohammed, N., Alobaid, F., Breuer, M., & Eppler, B. 2014. A comparative study on the influence of the gas flow rate on the hydrodynamics of a gas – solid spouted fluidized bed using Euler – Euler and Euler – Lagrange / DEM models. *Powder Technol. Elsevier B.V.* 264:343–364.

Almuntahar, A. & Taghipour, F. 2008. Computational fluid dynamics of high density circulating fluidized bed riser: Study of modeling parameters. *Powder Technol.* 185(1):11–23.

Alobaid, F., Almohammed, N., Massoudi, M., May, J., Philip, R., Richter, A., & Eppler, B. 2022. Progress in CFD Simulations of Fluidized Beds for Chemical and Energy Process Engineering. 91(October 2020).

Alobaid, F., Ströhle, J., & Eppler, B. 2013. Extended CFD / DEM model for the simulation of circulating fluidized bed. *Adv. Powder Technol. The Society of Powder Technology Japan.* 24(1):403–415.

Amblard, B. 2020. Use of CFD for the extrapolation of attrition and hydrodynamic phenomena in circulating fluidized beds. Université de Lyon.

Argyropoulos, C.D. & Markatos, N.C. 2015. Recent advances on the numerical modelling of turbulent flows. *Appl. Math. Model. Elsevier Inc.* 39(2):693–732.

Ariyaratne, W.K.H., Manjula, E.V.P.J., Ratnayake, C., & Melaaen, M.C. 2018. CFD Approaches for Modeling Gas-Solids Multiphase Flows - A Review. , in: The 9th EUROSIM Congress on Modelling and Simulation and The 57th SIMS Conference on Simulation and Modelling. Linköping University Electronic Press. . Linköping University Electronic Press, pp. 680–686.

Bandara, J.C., Thapa, R., Nielsen, H.K., Moldestad, B.M.E., & Eikeland, M.S. 2019. Circulating fluidized bed reactors – part 01: analyzing the effect of particle modelling parameters in

computational particle fluid dynamic (CPFD) simulation with experimental validation. *Part. Sci. Technol. Taylor and Francis Inc.* :1–14.

Belaid, M. 2017. Behaviour of selected SA coals in circulating fluidised bed (CFB) in comparison with russian coal. University of Witwatersrand.

Benzarti, S., Mhiri, H., & Bournot, H. 2012. Drag models for Simulation Gas-Solid Flow in the Bubbling Fluidized Bed of FCC Particles. *Waset.Org.* 61(1):1138–1143.

Berruti, F., Pugsley, T.S., Godfroy, L., Chaouki, J., & Patience, G.S. 1995. Hydrodynamics of circulating fluidized bed risers: A review. *Can. J. Chem. Eng.* 73(5):579–602.

Bing Du, M. 2005. Hydrodynamics and flow structure, gas and solids mixing behaviour, and choking phenomena in gas-solid fluidization. The Ohio State University.

Borhani, tohid N.G., Azarpour, A., & Akbari, V. 2015. CO₂ capture with potassium carbonate solutions : A state-of-the-art review. *Int. J. Greenh. Gas Control. Elsevier Ltd.* 41:142–162.

Cahyadi, A., Anantharaman, A., Yang, S., Karri, S.B.R., Findlay, J.G., Cocco, R.A., & Chew, J.W. 2017. Review of cluster characteristics in circulating fluidized bed (CFB) risers. *Chem. Eng. Sci. Elsevier.* 158(April 2016):70–95.

Cai, R., Ke, X., Lyu, J., Yang, H., Zhang, M., Yue, G., & Ling, W. 2017. Progress of circulating fluidized bed combustion technology in China : a review. *Clean Energy.* 1(1):36–49.

CD-Adapco. 2012. STAR-CCM+ user guide.

Chalermisinsuwan, B., Piumsomboon, P., & Gidaspow, D. 2010. A Computational Fluid Dynamics Design of a Carbon Dioxide Sorption Circulating Fluidized Bed. *AIChE J.* 56(11):2805–2824.

Chang, G., Song, C., & Wang, L. 2011. A modeling and experimental study of flue gas desulfurization in a dense phase tower. *J. Hazard. Mater. Elsevier B.V.* 189(1–2):134–140.

Chen, H., Zhou, X., & Feng, Z. 2021. Indoor and Built Application of polyhedral meshing strategy in indoor environment simulation: Model accuracy and computing time. *Indoor Built Environ.* 0(0):1–13.

Chen, J., Meng, C., Wang, S., Yu, G., Hu, T., & Lin, F. 2017. Effect of solid mass flux on anisotropic gas–solid flow in risers determined with an LES-SOM model. *Particuology. Chinese Society of Particuology.*

- Chen, X., Liu, J., Pang, Y., Chen, J., Chi, L., Gong, C., & Chen, X. 2020. Developing a new mesh quality evaluation method based on convolutional neural network neural network. *Eng. Appl. Comput. Fluid Mech. Taylor & Francis*. 14(1):391–400.
- Chen, X. & Wang, J. 2014. A comparison of two-fluid model, dense discrete particle model and CFD-DEM method for modeling impinging gas-solid flows. *Powder Technol. Elsevier B.V.* 254:94–102.
- Cocco, R., Karri, S.B.R., & Knowlton, T. 2014. Introduction to fluidization. *Chem. Eng. Prog.* 110(11):21–29.
- Córdoba, P. 2015. Status of Flue Gas Desulphurisation (FGD) systems from coal-fired power plants: Overview of the physic-chemical control processes of wet limestone FGDs. *Fuel*. 144:274–286.
- Cui, H., Sauriol, P., & Chaouki, J. 2003. High temperature uidized bed reactor : measurements , hydrodynamics and simulation. 58:1071–1077.
- Cui, L., Zhang, L., Li, B., Dong, Y., Yuan, X., & Ma, C. 2010. Effect investigation of composite humidification way on wall deposition of water droplets and desulfurization process in semi-dry CFB-FGD tower. *Asia-Pacific Power Energy Eng. Conf. APPEEC*. (2006).
- Dahiya, S., Anhäuser, A., Farrow, A., Thieriot, H., Chanchal, A., & Myllyvirta, L. 2020. Ranking the World's Sulfur Dioxide (SO₂) Hotspots: 2019-2020. Center for Research on Energy and Clean Air & Greenpeace India: Delhi.
- Dash, S., Mohanty, S., & Mishra, B.K. 2019. CFD modelling and simulation of an industrial scale continuous fluidized bed roaster. *Adv. Powder Technol. Elsevier B.V.*
- Davidson, J.F. 2000. Circulating fluidised bed hydrodynamics. *Powder Technol.* 113(3):249–260.
- Davidson, L. 2023. Fluid mechanics, turbulent flow and turbulence modelling [WWW Document]. Goteborg, Sweden. URL https://www.tfd.chalmers.se/~lada/postscript_files/solids-and-fluids_turbulent-flow_turbulence-modelling.pdf.
- Deng, Y., Ansart, R., Baeyens, J., & Zhang, H. 2019. Flue gas desulphurization in circulating fluidized beds. *Energies. MDPI AG*. 12(20).
- Department of Environmental Affairs and Tourism. 2004. National Environment Management: Air Quality Act (No. 39 of 2004). *Gov. Gaz.* 469(869):4–6.

- Ding, Y., He, Y., Cong, N.T., Yang, W., & Chen, H. 2008. Hydrodynamics and heat transfer of gas-solid two-phase mixtures flowing through packed beds - A review. *Prog. Nat. Sci.* 18(10):1185–1196.
- Du, J., Yue, K., Wu, F., Ma, X., & Hui, Z. 2021. Numerical investigation on the water vaporization during semi dry flue gas desulfurization in a three-dimensional spouted bed. *Powder Technol. Elsevier B.V.* 383:471–483.
- Dwivedi, K., Duttas, S., Loha, C., Karmakar, M., & Chatterjee, P. 2021. A numerical study on the wall erosion impact and gas-particle hydrodynamics in circulating fluidized bed riser. *Therm. Sci. Eng. Prog.* 22(100852).
- Ellendt, N., Lumanglas, A.M., Moqadam, S.I., & Mädler, L. 2018. A model for the drag and heat transfer of spheres in the laminar regime at high temperature differences. *Int. J. Therm. Sci. Elsevier.* 133(September 2017):98–105.
- Estrella, M.C.M. 2021. Electrostatic forces in fluidized bed reactors: numerical and experimental analysis. University de Toulouse.
- Fischer, M. & Darling, G. 2015. Circulating Fluidized Bed Scrubber vs. Spray Dryer Absorber [WWW Document]. *Power Eng Intern.* URL <https://www.power-eng.com/emissions/policy-regulations/circulating-fluidized-bed-scrubber-vs-spray-dryer-absorber>.
- Gao, X., Li, T., Sarkar, A., Lu, L., & Rogers, W.A. 2018. Development and Validation of an Enhanced Filtered Drag Model for Simulating Gas-Solid Fluidization of Geldart A Particles in All Flow Regimes. *Chem. Eng. Sci.*
- Gao, X., Zhong, Y., Wu, Z., Zhang, G., Evans, A., Jiang, M., Luo, Z., Ni, M., & Cen, K. 2010. Operating experience of CFB semi-dry FGD with novel humidification technology in China. *Air Waste Manag. Assoc. - 8th Power Plant Air Pollut. Control Mega Symp. 2010.* 2:1318–1358.
- Ghadirian, E., Abbasian, J., & Arastoopour, H. 2019. CFD simulation of gas and particle flow and a carbon capture process using a circulating fluidized bed (CFB) reacting loop. *Powder Technol. Elsevier B.V.* 344:27–35.
- Grace, J.R., Bi, H., & Golriz, M. 2003. Circulating Fluidised Beds. 1st ed. , in: Yang, W.-C. (Ed.), HANDBOOK of FLUIDIZATION and FLUID-PARTICLE SYSTEMS. CRC Press: Pittsburgh, USA. . CRC Press: Pittsburgh, USA, pp. 490–535.
- Grace, J.R. & Lim, C.J. 2013. Properties of circulating fluidized beds (CFB) relevant to

combustion and gasification systems. *Fluid. Bed Technol. Near-Zero Emiss. Combust. Gasif.* :149–176.

Guo, Y., Sun, J., Wang, R., Li, W., Zhao, C., & Li, C. 2021. Carbon Capture Science & Technology Recent advances in potassium-based adsorbents for CO₂ capture and separation : a review. *Carbon Capture Sci. Technol. Elsevier Ltd.* 1(October):100011.

Hartge, E.U., Ratschow, L., Wischnewski, R., & Werther, J. 2009. CFD-simulation of a circulating fluidized bed riser. *Particuology.* 7(4):283–296.

Helland, E., Bournot, H., Occelli, R., & Tadrist, L. 2007. Drag reduction and cluster formation in a circulating fluidised bed. *Chem. Eng. Sci.* 62(1–2):148–158.

Hiidenkari, H. 2018. Dynamic Core-Annulus Model of Circulating Fluidized Bed Boilers. Lappeenranta University of Technology.

Hill, F.F. & Zank, J. 2000. Flue gas desulphurization by spray dry absorption. *Chem. Eng. Process. Process Intensif.* 39(1):45–52.

Hodapp, M.J., Ramirez-behainne, J.J., Mori, M., & Jr, L.G. 2012. Numerical Studies of the Gas-Solid Hydrodynamics at High Temperature in the Riser of a Bench-Scale Circulating Fluidized Bed. 2012.

Horio, M. 2013. Overview of fluidization science and fluidized bed technologies. , Fluidized Bed Technologies for Near-Zero Emission Combustion and Gasification.

Hu, S. & Liu, X. 2020. A CFD-PBM-EMMS integrated model applicable for heterogeneous gas-solid flow. *Chem. Eng. J. Elsevier B.V.* 383.

Hussain, A., Ani, F.N., Darus, A.N., & Mustafa, A. 2005. EXPERIMENTAL AND COMPUTATIONAL STUDIES OF THE HYDRODYNAMICS OF THE RISER OF A CIRCULATING FLUIDIZED BED. , in: Brunei International Conference on Engineering and Technology.

Jiang, Y., Li, F., Ge, W., & Wang, W. 2020. EMMS-based solid stress model for the multiphase particle-in-cell method. *Powder Technol. Elsevier B.V.* 360(1):1337–1387.

Kaiser, S., Weigl, K., Spiess-Knafl, K., Aichernig, C., & Friedl, A. 2000. Modeling a dry-scrubbing flue gas cleaning process. *Chem. Eng. Process. Process Intensif.* 39(5):425–432.

Koech, L., Rutto, H., Lerotholi, L., Everson, R.C., Neomagus, H., Branken, D., & Moganelwa, A.

2021. Spray drying absorption for desulphurization: a review of recent developments. *Clean Technol. Environ. Policy. Springer Berlin Heidelberg*. 23(6):1665–1686.

Lary, L. De, Loschetter, A., Bouc, O., Rohmer, J., & Oldenburg, C.M. 2012. Assessing health impacts of CO₂ leakage from a geological storage site into buildings : Role of attenuation in the unsaturated zone and building foundation. *Int. J. Greenh. Gas Control. Elsevier Ltd*. 9:322–333.

Lateb, M., Masson, C., Stathopoulos, T., & Be, C. 2013. Comparison of various types of k – ϵ models for pollutant emissions around a two-building configuration. *J. Wind Energy Ind. Aerodyn*. 115:9–21.

Leung, D.Y.C., Caramanna, G., & Maroto-valer, M.M. 2014. An overview of current status of carbon dioxide capture and storage technologies. *Renew. Sustain. Energy Rev. Elsevier*. 39:426–443.

Leuschke, F. & Babcock, D. 2008. Flue gas desulphurization in a circulating fluidized bed : Investigation after 10 years of successful commercial operation at the facility of Pilsen/CZ. Germany.

Li, S., Tang, G., Wang, H., & Gao, H. 2011. Effect of Operating Conditions on the Efficiency of Circulating Fluidized Bed Flue Gas Desulphurization. , AISC.

Li, T., Gel, A., Pannala, S., Shahnam, M., & Syamlal, M. 2014. CFD simulations of circulating fluidized bed risers, part I: Grid study. *Powder Technol*. 254:170–180.

Li, Y., Li, F., & Qi, H. 2012. Numerical and experimental investigation of the effects of impinging streams to enhance Ca-based sorbent capture of SO₂. *Chem. Eng. J. Elsevier B.V*. 204–206:188–197.

Li, Y., Tong, H., Ma, C., Zhuo, Y., & Xu, X. 2004. Analytic study on approach to adiabatic saturation temperature and the control scheme for the amount of water sprayed in the semi-dry FGD process. *Fuel*. 83(17–18):2255–2264.

Liu, A.B., Mather, D., & Reitz, R.D. 1993. Modeling the Effects of Drop Drag and Breakup on Fuel Sprays. *Powder Technol*. 298:1–13.

Liu, J., Heidarinejad, M., Pitchurov, G., Zhang, L., & Srebric, J. 2018. An extensive comparison of modified zero-equation, standard k- ϵ , and LES models in predicting urban airflow. *Sustain. Cities Soc. Elsevier B.V*.

Loth, E. 2008. Drag of non-spherical solid particles of regular and irregular shape. *Powder*

Technol. 182(3):342–353.

Mani, F., Peruzzini, M., & Stoppioni, P. 2008. Combined Process of CO₂ Capture by Potassium Carbonate and Production of Basic Zinc (II) Carbonates : CO₂ Release from Bicarbonate Solutions at Room Temperature and Pressure. *Energy & Fuels*. 22:1714–1719.

Marocco, L. & Mora, A. 2013. CFD modeling of the Dry-Sorbent-Injection process for flue gas desulfurization using hydrated lime. *Sep. Purif. Technol. Elsevier B.V.* 108:205–214.

Miller, B.G. 2011. Emissions Control Strategies for Power Plants. *Clean Coal Eng. Technol.* :375–481.

Mohanty, C.R., Adapala, S., & Meikap, B.C. 2009. Removal of hazardous gaseous pollutants from industrial flue gases by a novel multi-stage fluidized bed desulfurizer. *J. Hazard. Mater.* 165(1–3):427–434.

Morakinyo, O.M., Mukhola, M.S., & Mokgobu, M.I. 2020. Ambient gaseous pollutants in an urban area in South Africa: Levels and potential human health risk. *Atmosphere (Basel)*. 11(7).

Moyeda, D., Newton, G., La Fond, J., Payne, R., & Kramlich, J. 2002. Rate controlling processes and enhancement strategies in humidification for duct SO₂ capture. NC.

Nagy, M.I., Csörgo, T., & Csanád, M. 2008. Detailed description of accelerating, simple solutions of relativistic perfect fluid hydrodynamics. *Phys. Rev. C - Nucl. Phys.* 77(2):1–15.

Neathery, J.K. 1996. Model for Flue-Gas Desulfurization in a Circulating Dry Scrubber. *AIChE J.* 42(1):259–268.

Neri, A. & Gidaspow, D. 2000. Riser Hydrodynamics : Simulation Using Kinetic Theory. *AIChE J.* 46(1).

Niewland, J.J., Huizenga, P., Kuipers, J.A.M., & van Swaaij, W.P.M. 1994. Hydrodynamic modelling of circulating fluidised beds. *Chem. Eng. Sci.* 49(24):5803–5811.

Nord, L.O. & Bolland, O. 2020. Carbon Dioxide Emission Management in Power Generation. Weinheim.

Olajire, A.A. 2010. CO₂ capture and separation technologies for end-of-pipe applications e A review. *EGY. Elsevier*. 35(6):2610–2628.

Panday, R., Breault, R., & Shadle, L.J. 2016. Dynamic modeling of the circulating fluidized bed riser. *Powder Technol. Elsevier*. 291:522–535.

Patience, G.S., Chaouki, J., Berruti, F., & Wong, R. 1992. Scaling considerations for circulating fluidized bed risers. , Powder Technology.

Philippsen, C.G., Vilela, A.C.F., & Zen, L.D. 2015. Fluidized bed modeling applied to the analysis of processes: Review and state of the art. *J. Mater. Res. Technol.* 4(2):208–216.

Pisani, R. & De Moraes, D. 2004. Removal of sulfur dioxide from a continuously operated binary fluidized bed reactor using inert solids and hydrated lime. *J. Hazard. Mater.* 109(1–3):183–189.

Poullikkas, A. 2015. Review of Design, Operating, and Financial Considerations in Flue Gas Desulfurization Systems. *Energy Technol. Policy.* 2(1):92–103.

Prix, R. 2004. Variational description of multifluid hydrodynamics: Uncharged fluids. *Phys. Rev. D - Part. Fields, Gravit. Cosmol.* 69(4):1–20.

Rahimi, A., Hatamipour, M.S., Gholami, M., & Haghnegahdar, M.R. 2011. Non-isothermal modeling of the flue gas desulphurization process using a semi-dry spouted bed reactor. *Chem. Eng. Res. Des.* 89(6):777–784.

Ramesh, P.L.N. & Raajenthiren, M. 2010. A Review of Some Existing Drag Models Describing the Interaction Between the Solid- Gaseous Phases in a CFB. *Int. J. Eng. Sci. Technol.* 2(5):1047–1051.

Rhodes, M. 2008. Introduction to Particle Technology. 2nd ed. John Wiley & Sons Inc.: West Sussex.

Sakaunnapaporn, C., Piumsomboon, P., & Chalermssinsuwan, B. 2017. Computational Fluid Dynamics Model of CO₂ Capture in Fluidized Bed Reactors : Operating Parameter Optimization. *Energy Procedia. Elsevier B.V.* 138:518–523.

Sanaei, S., Mostoufi, N., Radmanesh, R., Sotudeh-gharebagh, R., Guy, C., & Chaouki, J. 2010. Hydrodynamic Characteristics of Gas – Solid Fluidization at High Temperature. 88(February):1–11.

Shuai, W., Qi, W., Wang, S., Wang, Q., Chen, J., Liu, G., Lu, H., & Sun, L. 2016. Assessment of CO₂ capture using potassium-based sorbents in circulating fluidized bed reactor by multiscale modeling. *Fuel. Elsevier Ltd.* 164:66–72.

Siemens. 2017. NX-12 Help [WWW Document]. *Autom. Siemens.* URL https://docs.plm.automation.siemens.com/tdoc/nx/12/nx_help/#uid:index.

- Smolders, K. & Baeyens, J. 2001. Gas fluidized beds operating at high velocities : a critical review of occurring regimes. :269–291.
- Sommerfeld, M. & Huber, N. 1999. Experimental analysis and modelling of particle-wall collisions. *Int. J. Multiph. Flow.* 25:1457–1489.
- Sosnowski, M., Krzywanski, J., & Gnatowska, R. 2017. Polyhedral meshing as an innovative approach to computational domain discretization of a cyclone in a fluidized bed CLC unit. *E3S Web Conf. Energy Fuels.* 14(01027).
- Srivastava, R.K. & Jozewicz, W. 2001. Flue gas desulfurization: The state of the art. *J. Air Waste Manag. Assoc.* 51(12):1676–1688.
- Srivastava, R.K., Jozewicz, W., & Singer, C. 2001. SO₂ scrubbing technologies: A review. *Environ. Prog.* 20(4):219–228.
- Stroh, A., Alobaid, F., von Bohnstein, M., Ströhle, J., & Epple, B. 2018. Numerical CFD simulation of 1 MWth circulating fluidized bed using the coarse grain discrete element method with homogenous drag models and particle size distribution. *Fuel Process. Technol. Elsevier.* 169(September 2017):84–93.
- Taghipour, F., Ellis, N., & Wong, C. 2005. Experimental and computational study of gas-solid fluidized bed hydrodynamics. *Chem. Eng. Sci.* 60(24):6857–6867.
- Tao, M., Jin, B., Zhong, W., Yang, Y., & Xiao, R. 2010a. Numerical and experimental study on flue gas desulfurization in the underfeed circulating spouted bed. *Chem. Eng. J. Elsevier B.V.* 159(1–3):149–158.
- Tao, M., Jin, B., Zhong, W., Yang, Y., & Xiao, R. 2010b. Modeling and experimental study on multi-level humidifying of the underfeed circulating spouted bed for fl ue gas desulfurization. *Powder Technol. Elsevier B.V.* 198(1):93–100.
- Tritippayanon, R., Piemjaiswang, R., Piumsomboon, P., & Chalermssinsuwan, B. 2019. Computational fluid dynamics of sulfur dioxide and carbon dioxide capture using mixed feeding of calcium carbonate/calcium oxide in an industrial scale circulating fluidized bed boiler. *Appl. Energy. Elsevier.* 250(May):493–502.
- Tu, Q., Wang, H., & Ocone, R. 2022. Application of three-dimensional full-loop CFD simulation in circulating fluidized bed combustion reactors – A review. *Powder Technol. Elsevier B.V.* 399:117181.

- Uitbeijerse, A. 2011. Spray scrubbing for CO₂ capture at coal power plants. Delft University of Technology.
- Uwitonze, H., Kim, A., Kim, H., Brigljevic, B., Ly, H., Kim, S., Lim, H., & Upadhyay, H. 2022. CFD Simulation of hydrodynamics and heat transfer characteristics in gas-solid fluidized bed riser under fast pyrolysis flow condition. *Applied Therm. Eng.* 212(118555).
- Van Maele, K. & Merci, B. 2006. Application of two buoyancy-modified k – ϵ turbulence models to different types of buoyant plumes. *Fire Saf. J.* 41(2):122–138.
- Varas, A.E.C., Peters, E.A.J.F., & Kuipers, J.A.M. 2017. CFD-DEM simulations and experimental validation of clustering phenomena and riser hydrodynamics. *Chem. Eng. Sci. Elsevier Ltd.* 169:246–258.
- Versteeg, H., & Malalasekera, W. 2007. An introduction to Computational Fluid Dynamics: The finite volume method. 2nd editio ed. Pearson Education Limited.
- Vlavianos, C. 2020. South Africa's SO₂ emissions drop in 2019, no thank to government [WWW Document]. *Greenpeace*. URL <https://www.greenpeace.org/africa/en/press/12698/south-africas-so2-emissions-drop-in-2019-no-thanks-to-government/>.
- Wang, C. & Zhu, J. 2016. Developments in the understanding of gas-solid contact efficiency in the circulating fluidized bed riser reactor: A review. *Chinese J. Chem. Eng.* 24(1):53–62.
- Wang, N. & Zhang, X. 2011. Effect of humidification water on semi-dry flue gas desulfurization. *Procedia Environ. Sci.* 11(PART B):1023–1028.
- Wang, S., Li, X., Wu, Y., Li, X., Dong, Q., & Yao, C. 2010. Simulation of Flow Behavior of Particles in a Liquid – Solid Fluidized Bed Simulation of Flow Behavior of Particles in a Liquid - Solid Fluidized Bed. (March 2019).
- Wang, X., Wang, S., Wang, R., Yuan, Z., Shao, B., & Fan, J. 2021. Numerical simulation of semi-dry desulfurization spouted bed using the discrete element method (DEM). *Powder Technol. Elsevier B.V.* 378:191–201.
- Welty, J., Rogger, G., & Foster, D. 2015. Fundamentals of Momentum, Heat and Mass Transfer. 6th ed. John Wiley & Sons Inc.: Singapore.
- Wu, F., Bai, J., Yue, K., Gong, M., Ma, X., & Zhou, W. 2020. Eulerian-Eulerian Numerical Study of the Flue Gas Desulfurization Process in a Semidry Spouted Bed Reactor. *ACS Omega.* 5(7):3282–3293.

- Wu, G., Chen, W., & He, Y. 2020. Investigation on gas – solid flow behavior in a multistage fluidized bed by using numerical simulation. 364:251–263.
- Wu, Y., Peng, L., Qin, L., Wang, M., Gao, J., & Lan, X. 2018. Validation and application of CPFD models in simulating hydrodynamics and reactions in riser reactor with Geldart A particles. *Powder Technol. Elsevier B.V.* 323:269–283.
- Xie, J., Zhong, W., & Yu, A. 2018. MP-PIC modeling of CFB risers with homogeneous and heterogeneous drag models. *Adv. Powder Technol. The Society of Powder Technology Japan.* (August).
- Yalcin, S., Konukman, A.E.Ş., & Midilli, A. 2020. A perspective on fossil fuel based flue gas emission reduction technologies. *Greenh. Gases Sci. Technol.* 10(4):664–677.
- Yang, S. & Wang, S. 2020. Eulerian-Lagrangian simulation of the full-loop gas-solid hydrodynamics in a pilot-scale circulating fluidized bed. *Powder Technol. Elsevier B.V.* 369:223–237.
- Yates, J. 1996. EFFECTS OF TEMPERATURE AND PRESSURE ON GAS-SOLID FLUIDIZATION. *Chem. Eng. Sci.* 51(2):167–205.
- Yi, C., Jo, S., Seo, Y., Lee, J., & Ryu, C. 2007. Continuous operation of the potassium-based dry sorbent CO₂ capture process with two fluidized-bed reactors. 1:31–36.
- Yi, J., Sauer, H., Leuschke, F., & Babcock, D. 2005. What is possible to achieve on flue gas cleaning using the CFB technology? , in: 8th International Conference on Circulating Fluidized Beds. Hangzhou, China. . Hangzhou, China.
- Yin, S., Zhong, W., Jin, B., & Fan, J. 2014. Modeling on the hydrodynamics of pressurized high-flux circulating fluidized beds (PHFCFBs) by Eulerian-Lagrangian approach. *Powder Technol. Elsevier.* 259:52–64.
- Yoon, H., Stouffer, M.R., Rosenhoover, W.A., Withum, J.A., & Burke, F.P. 1988. Pilot process variable study of coalside desulfurization. *Environ. Prog.* 7(2):104–111.
- You, C. & Li, Y. 2013. Desulfurization characteristics of rapidly hydrated sorbents with various adhesive carrier particles for a semidry CFB-FGD system. *Environ. Sci. Technol.* 47(6):2754–2759.
- Zhang, H., Xiong, B., An, X., Ke, C., & Chen, J. 2020. Numerical prediction on the drag force and heat transfer of non-spherical particles in supercritical water. *Powder Technol. Elsevier B.V.*

361:414–423.

Zhang, J., You, C., Chen, C., Qi, H., & Xu, X. 2007. Effect of near-wall air curtain on the wall deposition of droplets in a semidry flue gas desulfurization reactor. *Environ. Sci. Technol.* 41(12):4415–4421.

Zhang, J., You, C., Zhao, S., Chen, C., & Qi, H. 2008. Characteristics and reactivity of rapidly hydrated sorbent for semidry flue gas desulfurization. *Environ. Sci. Technol.* 42(5):1705–1710.

Zhang, W. & You, C. 2016. Numerical simulation of particulate flows in CFB riser with drag corrections based on particle distribution characterization. *Chem. Eng. J. Elsevier B.V.* 303:145–155.

Zhang, X. & Wang, N. 2012. Experimental study on Fast Suspension Bed Flue Gas Desulphurization. *Energy Procedia.* 14:1665–1670.

Zhang, Y., Lan, X., & Gao, J. 2012. Modeling of gas-solid flow in a CFB riser based on computational particle fluid dynamics. *Pet. Sci.* 9(4):535–543.

Zhang, Y., Lei, F., & Xiao, Y. 2018. The influence of pressure and temperature on gas-solid hydrodynamics for Geldart B particles in a high-density CFB riser. *Powder Technol. Elsevier B.V.* 327:17–28.

Zhao, Q. & Zhang, Z. 2021. Application of Semi-dry Flue Gas Desulfurization Technology in Circulating Fluidized Bed Boilers. , in: 5th International Symposium on Resource Exploration and Environmental Science.

Zhou, Y., Peng, J., Zhu, X., & Zhang, M. 2011. Hydrodynamics of gas-solid flow in the circulating fluidized bed reactor for dry flue gas desulfurization. *Powder Technol.* 205(1–3):208–216.

Zhu, X., Dong, P., Tu, Q., Zhu, Z., Yang, W., & Wang, H. 2020. Investigation of gas-solids flow characteristics in a pressurised circulating fluidised bed by experiment and simulation. *Powder Technol.* 366:420–433.

ANNEXURE A CALIBRATION

In Annexure A, the calibration data of the different equipment (FAN-01, FDR-01) as well as the calibration curves are presented. For ease of process control, it was necessary to do these calculations prior to execution of the experiments. Furthermore, the calculations regarding the inlet gas temperature and relative humidity are also discussed in Annexure A.3.

A.1 Calibration data of the fan (FAN-01)

Since it was necessary to keep the inlet gas velocity constant throughout all of the experimental runs, the calibration of the fan was done prior to the experiments. On the control panel, the fan speed is controlled by inserting a percentage (%) value between 0-100%. However, the exact air velocity at these settings were unknown and had to be determined.

The procedure followed to do the calibration were as follows:

1. A hot-wire anemometer was used to measure the air velocity at the air inlet of the riser. The anemometer was placed so that the point of measure is in the centre of the inlet.
2. The fan was switched on with a setting of 10%.
3. The displayed measurements on the anemometer wireless handheld device checked until fluctuation were minimised.
4. When the fluctuations were at a minimum, velocity values were recorded on the handheld device at three (3) second intervals for a total of 60 seconds.
5. The average air velocity was automatically calculated and displayed on the handheld device, and this value was recorded for a fan setting of 10%.
6. Steps 3 - 5 were repeated, increasing the fan setting with 5% at a time and were done until the velocity at a fan setting of 100% has been recorded.

The air inlet velocity that were recorded during the procedure explained above were first multiplied with a correction factor that were determined to neglect the influence of the measuring point. The correction factor was determined by recording an average inlet velocity at different points across the area of the riser inlet. Equation A-1 has been used to calculate the correction factor:

$$\text{Velocity correction factor} = \frac{u_{\text{average}}}{u_{\text{measured},i}} = 1.06; i \text{ represents each measuring point} \quad (\text{A-1})$$

The corrected inlet velocity data is presented in Table A-1 with the calibration curve presented in Figure A-1.

Table A- 1: Calibration data of the fan for air inlet velocities

Fan speed (%)	Inlet velocity @ 200 mm ID (m/s)
0	0
10	2.067
15	3.2118
20	4.187
25	5.3318
30	6.3176
35	7.4306
40	8.5012
45	9.2856
50	10.1018
55	11.077
60	11.7978
65	12.9638
70	14.0556
75	14.7976
80	15.9636
85	16.536
90	17.596
95	18.656
100	19.5358

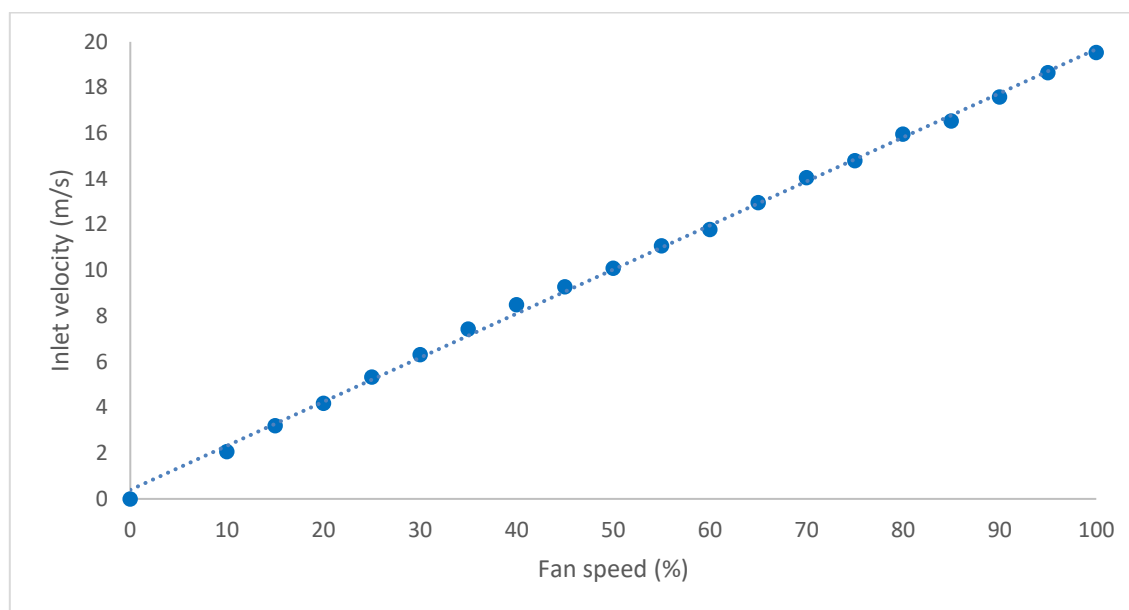


Figure A- 1: Calibration curve of the fan

A.2 Calibration data of the sorbent feeder (FDR-01)

In this study, the sorbent mass flow rate was fixed for all off the experimental runs. It was therefore necessary to determine the mass flow rate (kg/s) for a range of feeder settings (%). The following procedure were followed to do the feeder calibration:

1. First the feeder and the mixer were switched on and the feeder setting was set to 10%.
2. A container was placed under the feeder outlet to capture the sorbent discharge used during calibration for re-use. The container was weighed and the initial weight was recorded.
3. The hopper was loaded and with the use of a stopwatch, the sorbent discharge were collected for 60s.
4. After 60s, the container was replaced and the final weight of the container were determined. The total sorbent weight collected during the 60s were calculated and also recorded.
5. The sorbent mass flow rate was determined dividing the total sorbent weight by 60s.
6. Steps 2 - 5 were repeated, increasing the feeder setting 5% each time until the flow rate at a feeder setting of 100% can be determined.

In Table A-2, the data recorded for hydrated lime during the calibration procedure, as well as the calculated mass flow rate (kg) are reported with the calibration data for potassium carbonate in Table A-3.

Table A- 2: Hydrated lime feeder calibration data

Feeder speed (%)	Time (s)	Initial weight (kg)	Final weight (kg)	Sorbent weight (kg)	Flow rate (kg/s)
10	60	3.0	3.6	0.6	0.01
15	60	5.4	6.6	1.2	0.02
20	60	3.2	5	1.8	0.03
25	60	3.4	5.8	2.4	0.04
30	60	3.2	6.2	3	0.05
35	60	3.2	6.8	3.6	0.06
40	60	3.2	7.4	4.2	0.07
45	60	3.2	8	4.8	0.08
50	60	3.2	8.6	5.4	0.09
55	60	3.2	8.6	5.4	0.09
60	60	3.2	9.2	6	0.10
65	60	3.2	9.8	6.6	0.11
70	60	3.2	10.4	7.2	0.12
75	60	3.2	10.4	7.2	0.12
80	60	3.2	11	7.8	0.13
85	60	3.2	11.6	8.4	0.14
90	60	3.2	12.2	9	0.15
95	60	3.2	12.2	9	0.15
100	60	3.2	12.8	9.6	0.16

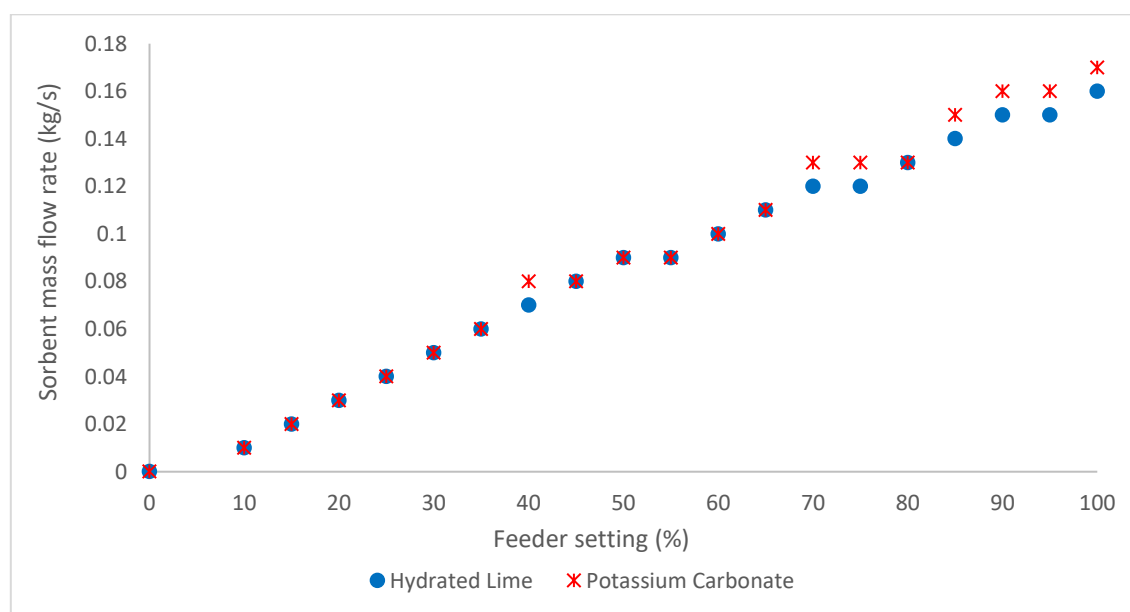
Table A- 3: Potassium Carbonate feeder calibration data

Feeder speed (%)	Time (s)	Initial weight (kg)	Final weight (kg)	Sorbent weight (kg)	Flow rate (kg/s)
10	60	3	3.4	0.4	0.01
15	60	5.4	6.5	1.1	0.02
20	60	3.2	5.1	1.9	0.03
25	60	3.4	5.7	2.3	0.04
30	60	3.2	6.4	3.2	0.05
35	60	3.2	6.9	3.7	0.06
40	60	3.2	7.7	4.5	0.08
45	60	3.2	8.2	5	0.08
50	60	3.2	8.7	5.5	0.09
55	60	3.2	8.8	5.6	0.09
60	60	3.3	9.5	6.2	0.10
65	60	3.4	10.2	6.8	0.11
70	60	3.2	10.7	7.5	0.13
75	60	3.2	10.7	7.5	0.13
80	60	3.2	11	7.8	0.13

85	60	3.2	11.9	8.7	0.15
90	60	3.2	12.7	9.5	0.16
95	60	3.2	12.9	9.7	0.16
100	60	3.2	13.3	10.1	0.17

In Figure A-3, the feeder calibration curve for both hydrated lime and potassium carbonate are illustrated.

Figure A- 2: Feeder calibration curve for hydrated lime and potassium carbonate



A.3 Inlet gas temperature and relative humidity calculations

The method of humidifying the gas (air) stream and controlling the temperature required calculations to determine the water flow rate through the nozzle. The nozzles were located downstream of the air-preheater, and upstream of the riser inlet. Furthermore, the calculations had to consider the ambient conditions of each day, since the facility was exposed to environmental conditions. The calculations were checked each day to correct the water flowrate that were required for the conditions under which the experiment will be performed for the day. The air properties that were considered in the calculations are reported in Table A-4.

Table A- 4: Air properties

Property	Value	Unit
Heat Capacity (C_p)	0.029	kJ/mol.K
Density of air	1.08	kg/m ³
Molecular weight (MW)	29	g/mol

Initial temperature (T)	*Varying	°C
Final temperature (T)	60 / 70 / 80	°C
Volumetric flow rate (Q)	Max = 1260	m ³ /h

From psychometric charts, the dry-bulb temperature and relative humidity for each respective day was used to determine the dew point temperature (°C) and the absolute humidity (kg water/kg dry air). Furthermore, the water content of the air was calculated, using the mass flow of air multiplied by the absolute humidity.

Next, the water content of the desired air relative humidity was calculated using the psychometric charts. The water content in kg water/kg dry air are reported in Table A-5.

Table A- 5: Water content at set point relative humidity at 1 bar (kg water/kg DA)

Target Relative Humidity	Final Temperature (°C)		
	60	70	80
10%	0.013	0.0200	0.03
20%	0.026	0.041	0.064
30%	0.039	0.063	0.1

Using the air properties, and the values from Table A-5, the required water flow rate was determined. For the purpose of presenting values, it can be assumed that the inlet air had the following properties:

- Inlet Temperature: 25°C
- Air mass flow rate: 1260 kg/hr
- Dew point: 5.3°C
- Absolute humidity: 0.01 kg water/kg DA
- Water content: 12.60 kg/hr

The required water flow rate to be added to the air stream for achieving the target relative humidity are presented in Table A-6.

Table A- 6: Water flow rate (kg/hr) required to achieve set point relative humidity at 1 Bar

Target Relative Humidity	Final Temperature (°C)		
	60	70	80

10%	3.62	12.35	24.82
20%	19.83	38.54	67.23
30%	36.05	65.99	112.14

A new burner was required and therefore, all of the above values were used to determine the energy output required from the burner to increase the air temperature from the initial to the final state, while ensuring that all water droplets is evaporated into the air stream to reach the target relative humidity. Furthermore, a burner efficiency of 80% is assumed for the new burner, and the true energy requirement could be calculated.

Table A- 7: Power requirements for various air velocities

Air velocity (m/s)	Air flow rate (m³/hr)	Air flow rate (kg/hr)	Water content (kg/hr)	Power air (kW)	Water added (kg/hr)	Power Water (kW)	Total Power (kW)	Actual Power (kW)
1.95	220.54	238.18	2.38	2.32	0.68	0.46	2.78	2.22
3.03	342.68	370.10	3.70	3.60	1.06	0.72	4.32	3.45
3.95	446.73	482.47	4.82	4.69	1.38	0.94	5.63	4.50
5.03	568.88	614.39	6.14	5.97	1.76	1.20	7.17	5.74
5.60	633.35	684.01	6.84	6.65	1.96	1.33	7.98	6.39
7.01	792.81	856.24	8.56	8.32	2.46	1.67	9.99	7.99
8.02	907.04	979.60	9.80	9.52	2.81	1.91	11.43	9.14
8.76	990.73	1069.99	10.70	10.40	3.07	2.08	12.49	9.99
9.53	1077.82	1164.04	11.64	11.32	3.34	2.27	13.58	10.87
10.45	1181.87	1276.42	12.76	12.41	3.66	2.48	14.89	11.92
11.13	1258.77	1359.48	13.59	13.22	3.90	2.65	15.86	12.69
12.23	1383.18	1493.83	14.94	14.52	4.29	2.91	17.43	13.95
13.26	1499.67	1619.64	16.20	15.75	4.65	3.15	18.90	15.12
13.96	1578.84	1705.15	17.05	16.58	4.89	3.32	19.90	15.92
15.06	1703.25	1839.51	18.40	17.88	5.28	3.58	21.46	17.17
15.60	1764.32	1905.46	19.05	18.53	5.47	3.71	22.23	17.79
16.60	1877.42	2027.61	20.28	19.71	5.82	3.95	23.66	18.93
17.60	1990.51	2149.75	21.50	20.90	6.17	4.18	25.09	20.07
18.43	2084.38	2251.13	22.51	21.89	6.46	4.38	26.27	21.01

In Table A.7, the power requirements were calculated for a wide range of air velocities. The calculations were based on all of the previously mentioned air properties and additionally, the following properties were considered:

- Initial air temperature: 25°C
- Final air temperature: 60°C
- Final air relative humidity: 10%
- Water vaporization heat: 2441.7 kJ/kg
- Water heat capacity: 4.81 kJ/kg.K
- Water molecular weight: 18 g/mol

However, the properties above were changed for the other experimental conditions. The water flow rate was determined accordingly through two nozzles and manual measurements of the air temperature and relative humidity before the riser inlet were taken to confirm the required experimental conditions.

A.4 Inlet gas temperature and relative humidity measurements

Table A- 8: Manual inlet gas temperature and relative humidity measurements

Operating conditions to be reached:		Measured variable	Section			
Temperature (°C)	%RH		Inlet	1	2	3
60	10	T (°C)	61.62	59.62	59.12	58.32
		RH (%)	13%	10%	11%	11%
	20	T (°C)	61.78	59.78	59.28	58.48
		RH (%)	21%	22%	20%	20%
	30	T (°C)	61.05	59.05	58.55	57.75
		RH (%)	30%	30%	31%	31%
70	10	T (°C)	70.63	69.83	69.25	69.58
		RH (%)	12%	11%	11%	10%
	20	T (°C)	70.54	71.23	69.94	70.55
		RH (%)	22%	21%	22%	20%
	30	T (°C)	71.89	69.56	68.98	68.04
		RH (%)	31%	30%	31%	31%
80	10	T (°C)	79.01	79.44	78.83	77.76
		RH (%)	11%	10%	11%	11%
	20	T (°C)	80.58	79.15	78.53	78.91
		RH (%)	20%	20%	20%	20%

30	T (°C)	80.72	79.81	79.14	79.52
	RH (%)	31%	31%	30%	31%

ANNEXURE B EQUIPMENT AND INSTRUMENTATION

B.1 General Information

During the execution of experimental runs several different equipment and instrumentation were used to either set fixed operating parameters or measure depending variables. The applicable equipment are listed below along with its operating range and the make and model.

Table B- 1: Measurement instrument and process unit information

Equipment/ Instrumentation	Operating range	Accuracy	Supplier/ Model
Variable speed fan	0.00 – 18.43 m/s	$\pm (0.03 + (0.04 \times U))$	Trojan fans/ BC10
Variable speed screw feeder	0.00 – 0.34 kg/s	± 0.01 kg/s	ZEST Weg/ MBF with agitator
Flush pressure transmitters	0.0 – 10 kPa	± 0.05 %	Wika/ S-11
Resistance temperature detectors	0.0 – 300.0 °C	$\pm 0.6^{\circ}\text{C}$	Swift heat & control/ PT-100
Hot wire anemometer	0.0 – 50.0 m/s	$\pm (0.03 + (0.04 \times U))$	Testo/ Testo 440
Oil burner	30-60 kW	-	Unigas LO60 Burner
Vacuum pump	Airflow: 23 L/min Vacuum: 600 mmHg	± 0.05 %	Aircraft/ SG VAC05
High pressure nozzles	-	-	Delavan Oil burner nozzles/ OB20.00 – 60B

B.2 Sorbent sampling procedure

To be able to determine the radial and axial solids distribution across the riser, it was necessary to have real-time data from the experiments. A non-isokinetic sampler as illustrated in Section 3.4.3, was designed and used to extract solids from specific points in the riser.

The non-isokinetic sampler consisted of the following parts:

- Perspex collection bucket
- Sampling port screw lid
- Toggle valve (open/close valve)

The following procedure were followed during sampling of the sorbent particles:

1. The perspex collection container were weighed together with the filter paper disc. The weight was recorded.
2. The sampler was assembled, placing the filter paper in the sampling container, then the Perspex bucket were put into place after which the sampler were assembled.
3. Prior to sampling, the desired suction velocity was determined using the Testo 440 hot wire anemometer and the vacuum pump was set accordingly. This made the control of the sampler very easy. The sampling procedure was only executed once the riser has reached steady state.
4. The vacuum pump was switched on, while the toggle valve on the sampler were in a closed position.
5. Once steady state has been reached, the bottom port was opened and the sampler was connected to the riser with the sampling port screw lid. This ensured that the sampling port was sealed.
6. The sampler was positioned to take a sorbent sample at the first radial position with the sampling tip facing downwards.
7. A stopwatch was used to record 120 seconds as soon as the toggle valve was opened. After 120 seconds, the toggle valve was closed again.
8. The sampler was disassembled, while the sampler probe remained connected to the riser.
9. The sorbent sample collected, together with the perspex bucket and filter paper were weighed, and the weight was recorded.

10. The perspex bucket was cleaned and a new filter paper is used.
11. The sampler was assembled and moved to the second radial position.
12. Steps 7-11 is repeated for all the radial positions.
13. After the bottom port sampling has been completed, the sampler is removed and the whole procedure is repeated again for the middle port and the top port.
14. After the sampling at each port, the suction velocity is measured and confirmed to resemble the desired suction velocity as determined.

ANNEXURE C CONFIDENCE INTERVAL CALCULATIONS

Confidence intervals for the experimental runs were calculated to assess the repeatability of the measurements. Data from four experimental runs with the same conditions has been used to in the calculations.

C.1 Hydrated Lime (Ca(OH)_2) confidence interval

Four experimental runs were executed with the same experimental conditions. In this case, the repeatability was assessed for an inlet gas temperature of 60°C and 10% inlet relative humidity.

In Table C-1, the respective solids volume fractions for the four experimental runs are reported.

Table C- 1: Ca(OH)_2 – Experimental radial solids volume fractions at each radial position for the different riser ports

Port	r/R	Experimental runs			
		1	2	3	4
Bottom	-1	2.25E-04	2.63E-04	2.13E-04	2.75E-04
	0	2.11E-04	2.25E-04	2.08E-04	2.16E-04
	1	2.51E-04	2.83E-04	2.40E-04	3.57E-04
Middle	-1	1.95E-04	2.28E-04	1.84E-04	2.19E-04
	0	1.77E-04	1.82E-04	1.74E-04	1.81E-04
	1	2.65E-04	3.36E-04	2.48E-04	3.50E-04
Top	-1	2.72E-04	3.45E-04	2.63E-04	3.87E-04
	0	1.83E-04	1.96E-04	1.78E-04	2.05E-04
	1	1.75E-04	1.99E-04	1.65E-04	2.14E-04

The data analysis function of Excel was used to determine the mean and 95% Confidence Interval (CI). In Table C-2, the sample mean and confidence interval for each radial position and riser ports is given.

Table C- 2: Ca(OH)_2 – Calculated sample mean and CI (95%) for each radial position at the different riser ports

Statistic	Radial Position (r/R)
-----------	-----------------------

Riser port		-1	0	1
Bottom	Mean:	2.44E-04	2.15E-04	2.83E-04
	CI (95%)	4.71E-05	1.22E-05	8.40E-05
Middle	Mean:	2.07E-04	1.79E-04	3.00E-04
	CI (95%)	3.22E-05	5.45E-06	8.11E-05
Top	Mean	3.17E-04	1.90E-04	1.88E-04
	CI (95%)	9.48E-05	1.93E-05	3.52E-05

In Figure C-1, C-2 and C-3 the average solids volume fraction and the confidence interval for each radial position is illustrated. In Figure C-1, the confidence intervals obtained near the wall of the riser is much higher, than in the centre of the riser. Again, in Figure C-2, the calculated confidence interval for the middle port, is the largest near the wall of the riser. This would indicate a greater degree of uncertainty near the wall of the riser. The same is also applicable in for the top riser, illustrated in Figure C-3. However, the radial position $r/R = -1$ has the largest confidence interval, while the bottom and middle ports has the largest confidence interval at radial position $r/R = 1$.

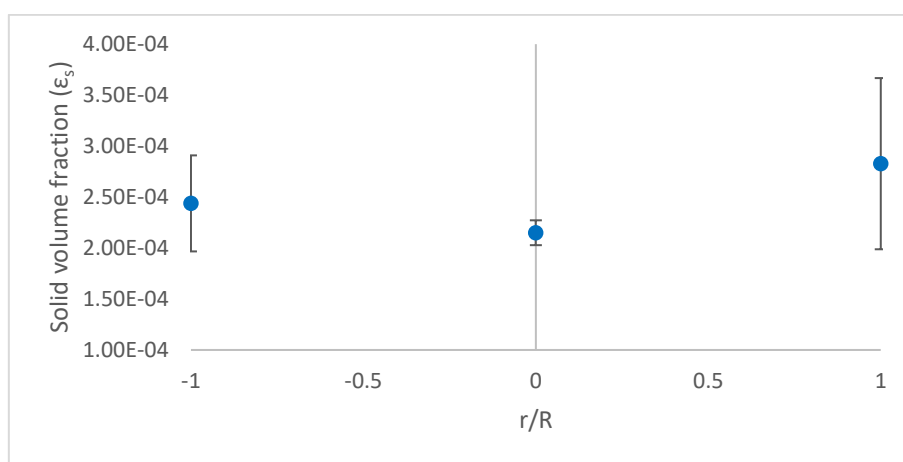


Figure C- 1: Ca(OH)_2 – Average solids volume fraction with associated confidence intervals at the bottom port of the riser

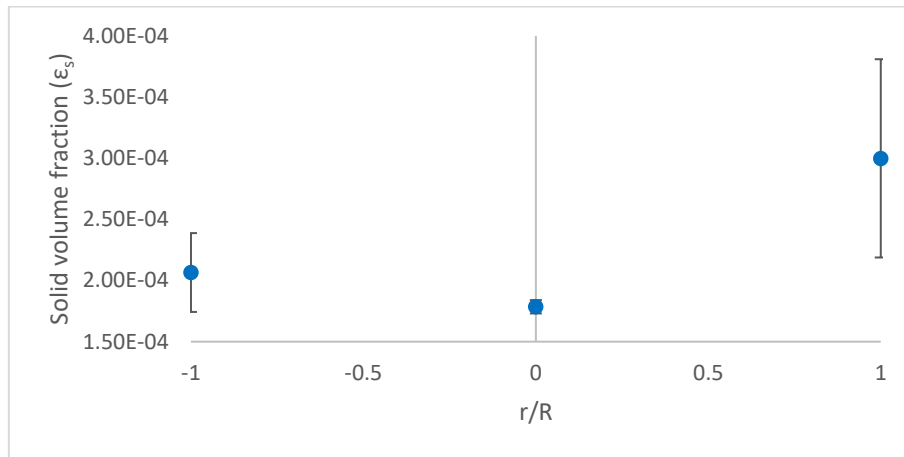


Figure C- 2: Ca(OH)_2 – Average solids volume fraction with associated confidence intervals at the middle port of the riser

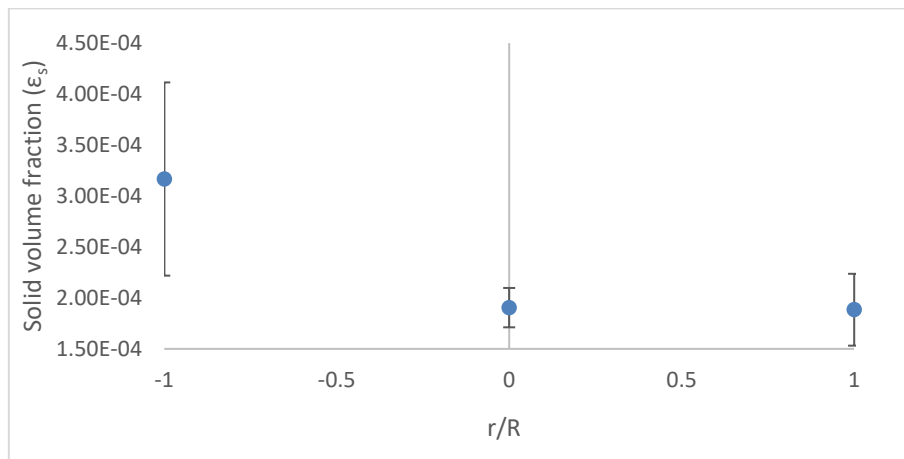


Figure C- 3: Ca(OH)_2 – Average radial solids volume fraction with associated confidence intervals at the top port of the riser

Wall friction inter-particle interactions can be the main contributors to the large variation in solids volume fractions near the wall. These were also discussed in Chapter 5, as effects that influence the radial distributions and particle accumulation near the wall. However, the large variations are not considered crucial for the purpose of this study, but rather as a statistical indicator on the continuous performance of the facility and executed experiments. In Figure C-4, the confidence interval for the average axial solids fraction associated with the different riser ports are illustrated. It can be seen that the confidence interval for the top and bottom port are very similar, while that of the middle port is slightly smaller which indicated a higher degree of certainty compared to the other ports. the higher confidence interval at the top port is likely due to recirculation in the top part of the riser, while the bottom port higher confidence interval can be ascribed to the change in riser geometry from the cone to the upper region causing flow to be disturbed.

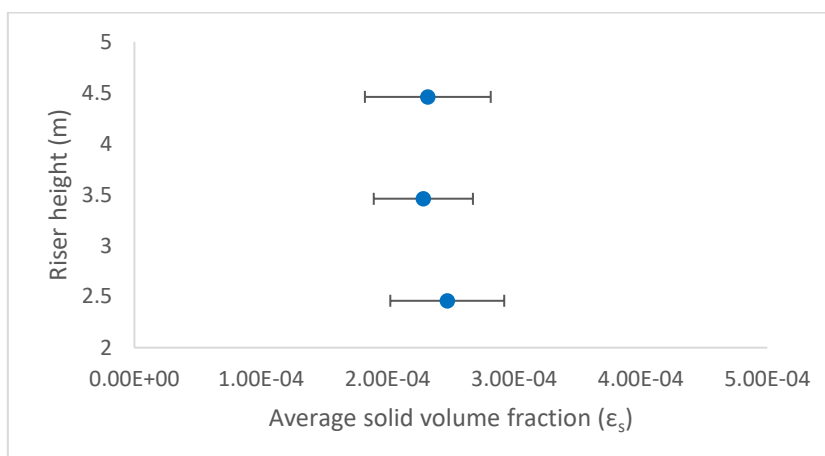


Figure C- 4: Ca(OH)_2 – Average axial solids volume fraction with associated confidence intervals at the different riser ports

The confidence interval associated with temperature in the riser at the respective temperature measuring points is illustrated in Figure C-5. It is seen that as the height increases the confidence interval also increases. The smallest confidence interval is associated with the inlet temperature, which is expected, as this is the set point of each experimental run. Although large variation in certainty exist for the temperature from T1 to T3, it is not considered to be influential for the purpose of this study.

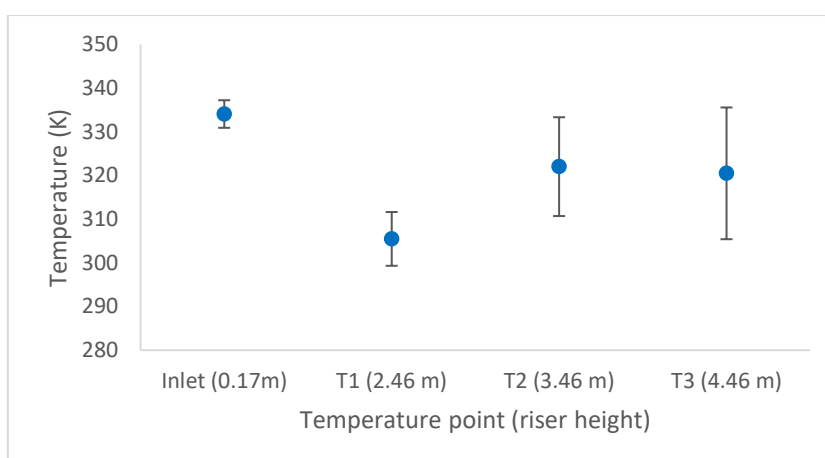


Figure C- 5: Ca(OH)_2 – Temperature mean and associated confidence intervals across the riser

Finally, the confidence intervals associated with the two pressure differential reported in this study are given in Table C-3. It is seen that the confidence interval for the pressure differential across the riser are much higher to that of the pressure differential across the upper region of the riser. The uncertainty for the pressure differential across the riser is likely due to the large pressure drop due to the venturi, and cluster formation and deformation in the riser. However, the confidence intervals are not that large in value and the variation was not considered for the purpose of this study.

Table C- 3: Ca(OH)_2 – Pressure differential mean and associated confidence interval across the riser and upper region

	Mean	CI (95%)
Riser	130.99	10.30
Upper region	9.94	1.17

C.2 Potassium carbonate (K_2CO_3) confidence interval

The same procedure has been followed for the determination of the confidence interval for various properties when potassium carbonate is used as sorbent. Four experimental runs have been performed with identical operating conditions. In Table C-4, the solids volume fraction at different radial positions is reported. These values have been used in the analysis. Excel data analysis function for descriptive statistics has been used to calculate the mean and 95% confidence interval.

Table C- 4: K_2CO_3 - Experimental radial solids volume fractions at each radial position for the different riser ports

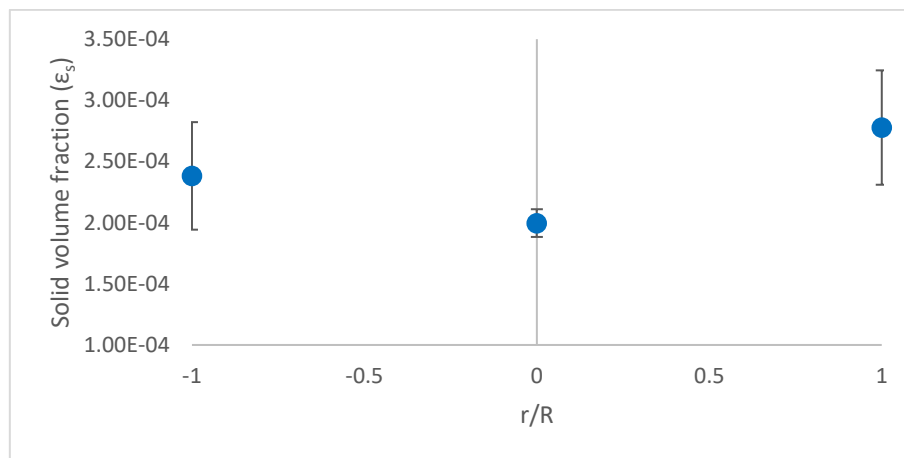
Port	r/R	Experimental runs			
		1	2	3	4
Bottom	-1	2.21E-04	2.58E-04	2.09E-04	2.65E-04
	0	1.96E-04	2.09E-04	1.93E-04	2.00E-04
	1	2.59E-04	2.93E-04	2.48E-04	3.11E-04
Middle	-1	2.05E-04	2.31E-04	1.94E-04	2.24E-04
	0	1.57E-04	1.62E-04	1.55E-04	1.58E-04
	1	2.96E-04	3.61E-04	2.83E-04	3.50E-04
Top	-1	2.86E-04	3.34E-04	2.76E-04	3.78E-04
	0	1.76E-04	1.80E-04	1.75E-04	1.97E-04
	1	1.85E-04	2.10E-04	1.75E-04	2.20E-04

From the analysis in Excel, the mean and CI (95%) values as retrieved are reported in Table C- 5. In Figure C-6, C-7 and C-8 the mean solids volume fractions with associated confidence intervals is illustrated.

Table C- 5: K_2CO_3 - Calculated sample mean and CI (95%) for each radial position at the different riser ports

Riser port	Statistic	Radial Position (r/R)		
		-1	0	1
Bottom	mean:	2.38E-04	2.00E-04	2.78E-04
	CL	4.39E-05	1.13E-05	4.68E-05
Middle	mean:	2.14E-04	1.58E-04	3.23E-04
	CL	2.74E-05	4.30E-06	6.14E-05
Top	mean	3.19E-04	1.82E-04	1.97E-04
	CI	7.50E-05	1.63E-05	3.40E-05

It is seen that for all three ports the confidence intervals near the riser wall are significantly larger compared to the confidence interval at the centre of the riser. The small confidence interval at the centre indicates a high level of certainty in that region, while the level of certainty near the wall is lower. It is also seen that at the bottom and middle ports the confidence interval at radial position $r/R = 1$, are the highest, while at the top port the confidence interval at radial position $r/R = -1$ are the highest.

Figure C- 6: K_2CO_3 - Average solids volume fraction with associated confidence intervals at the bottom port of the riser

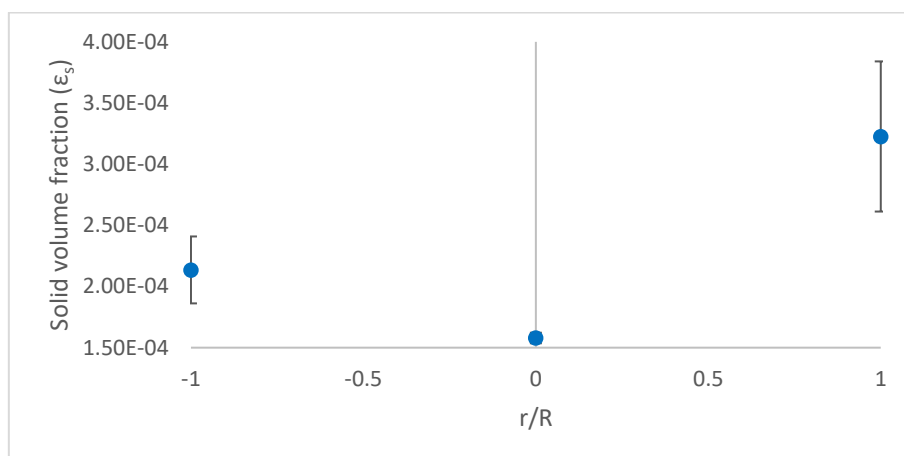


Figure C- 7: K_2CO_3 - Average solids volume fraction with associated confidence intervals at the middle port of the riser

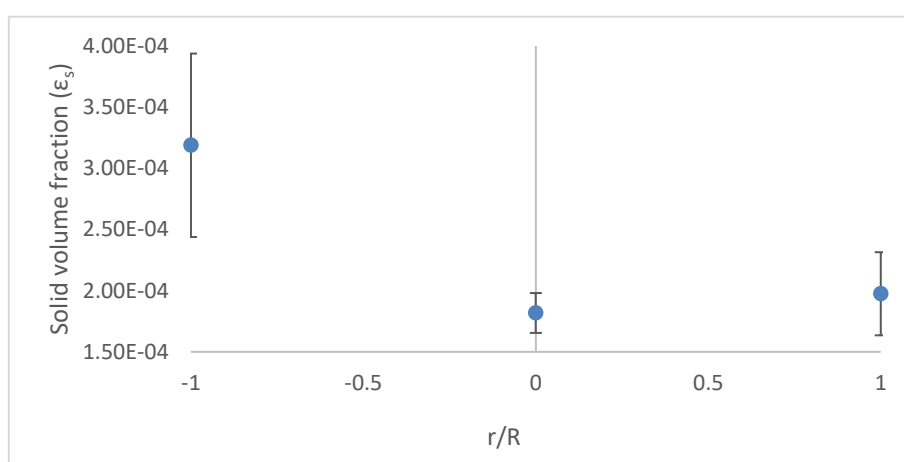


Figure C- 8: K_2CO_3 - Average solids volume fraction with associated confidence intervals at the top port of the riser

Although the confidence interval is a good indication of the level of certainty when doing measurements during experimental runs, the variation in this particular study was not considered to be significant due to the order of magnitude. In Figure C-9, the average axial solids distribution together with the associated confidence intervals are illustrated. From the figure, it is seen that the top port has the highest confidence interval, while the middle and bottom port has a very similar confidence interval. The higher variation in data at the top port can be ascribed to the recirculation in the top region of the riser.

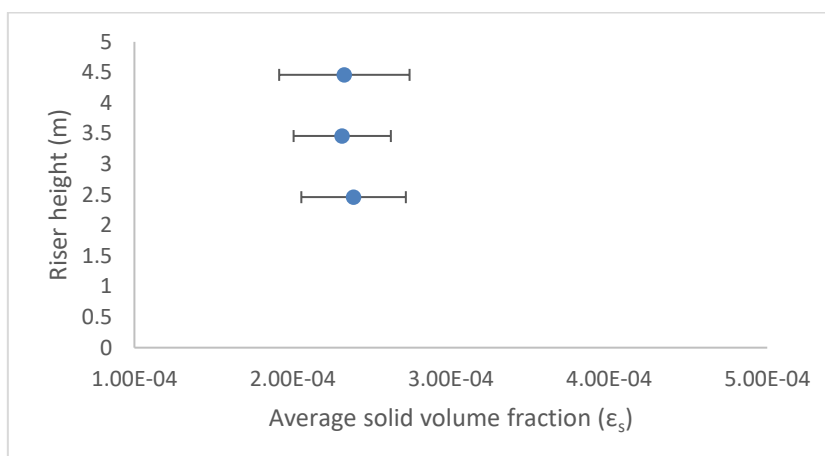


Figure C- 9: K_2CO_3 - Average axial solids volume fraction with associated confidence intervals at the different riser ports

The confidence interval associated with the temperature in the riser are illustrated in Figure C-10. The smallest confidence interval is reported to be at the inlet, which is expected as the inlet temperature is also the set point temperature. Furthermore, the temperature at T2 shows the highest level of uncertainty, with T3 the lowest in the upper region of the riser.

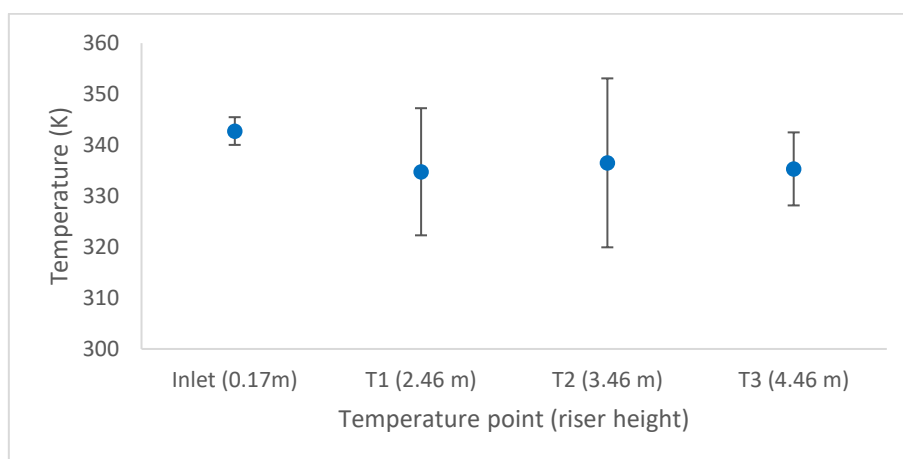


Figure C- 10: K_2CO_3 - Temperature mean and associated confidence intervals across the riser

Finally, the mean and confidence interval for the two respective pressure differentials addressed in this study are reported in Table C-6. It is seen that the confidence interval for the pressure differential is almost four times bigger in magnitude compared to that of the pressure differential across the upper region.

Table C- 6: K_2CO_3 - Pressure differential mean and associated confidence interval across the riser and upper region

	Mean	CI (95%)
Riser	153.24	5.13
Upper region	17.53	1.51

ANNEXURE D HYDRATED LIME ADDITIONAL RESULTS

D.1 Experimental radial solids distribution for hydrated lime

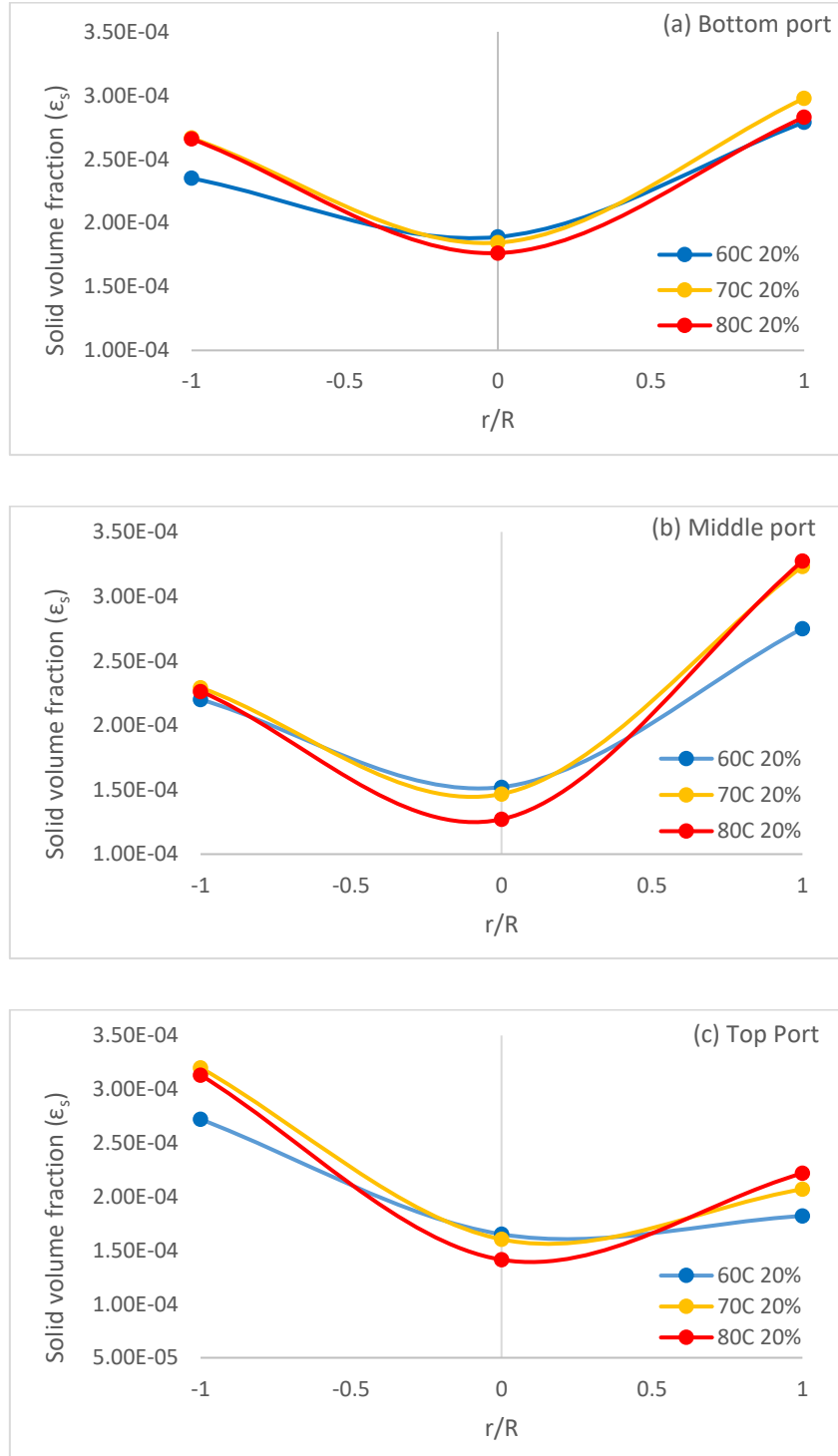


Figure D- 1: Ca(OH)_2 – Experimental radial solids distribution profile at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas temperature and 20% RH

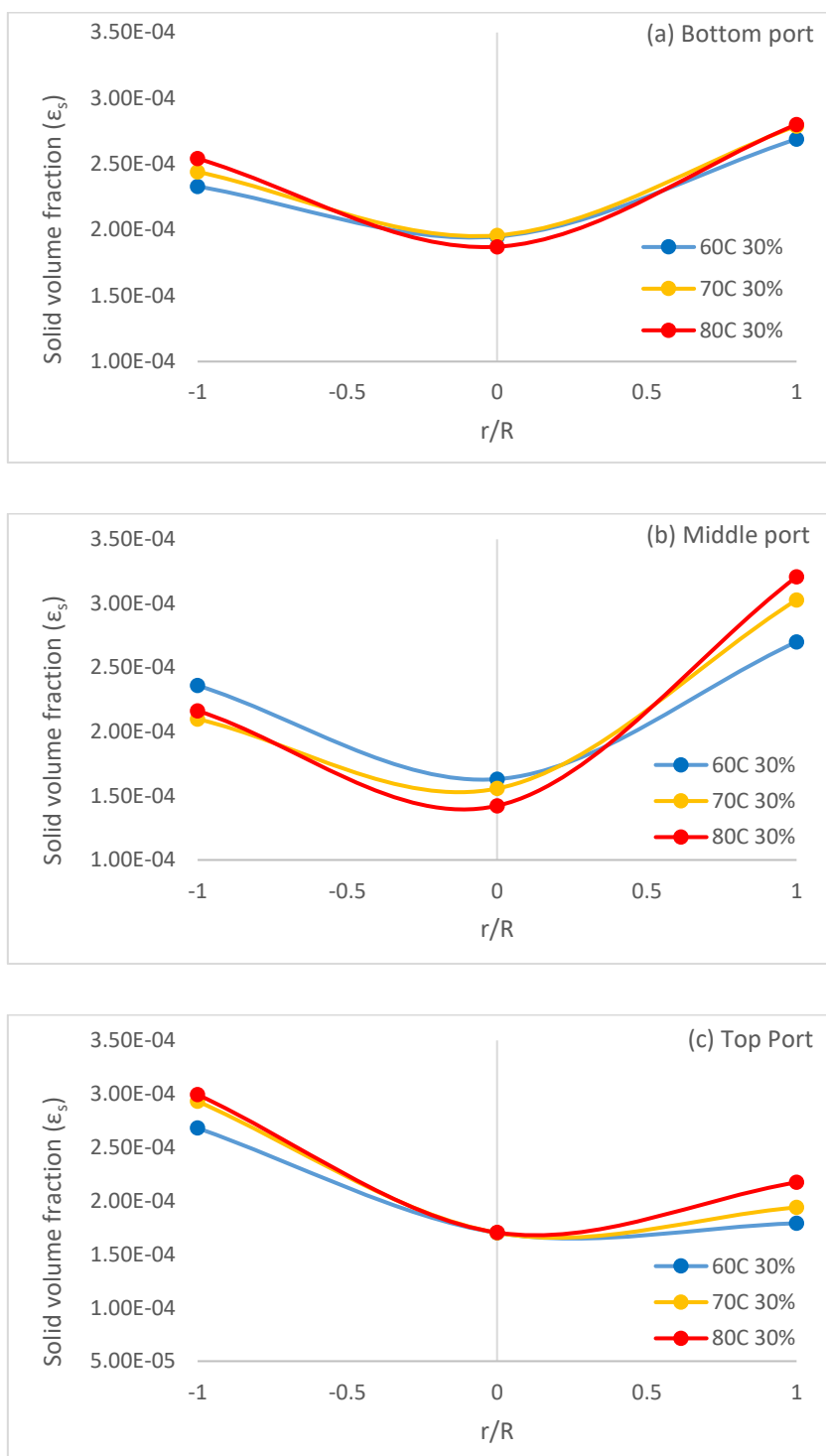


Figure D- 2: Ca(OH)_2 – Experimental radial solids distribution profile at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas temperature and 30% RH

D.2 Modelling radial distribution for hydrated lime

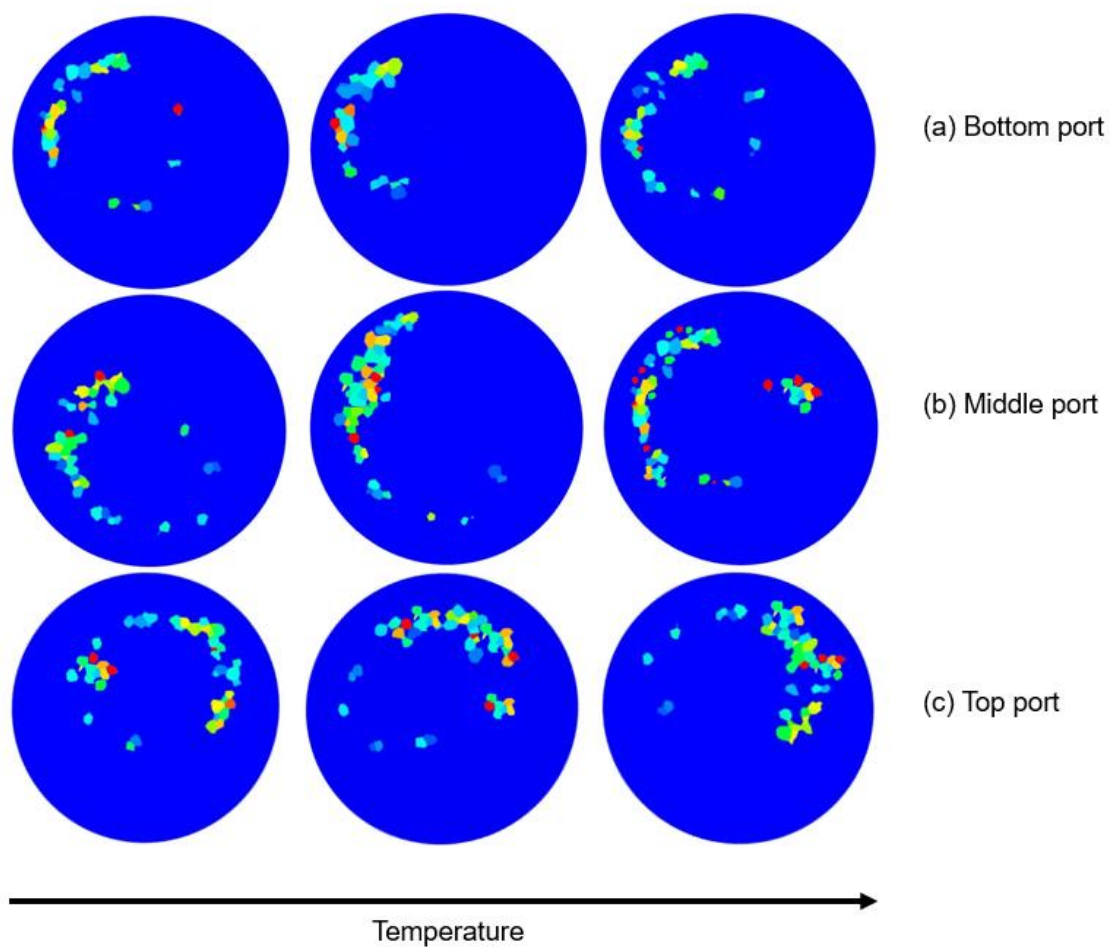


Figure D- 3: Ca(OH)_2 – Modelled radial solids volume fraction at (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas temperature

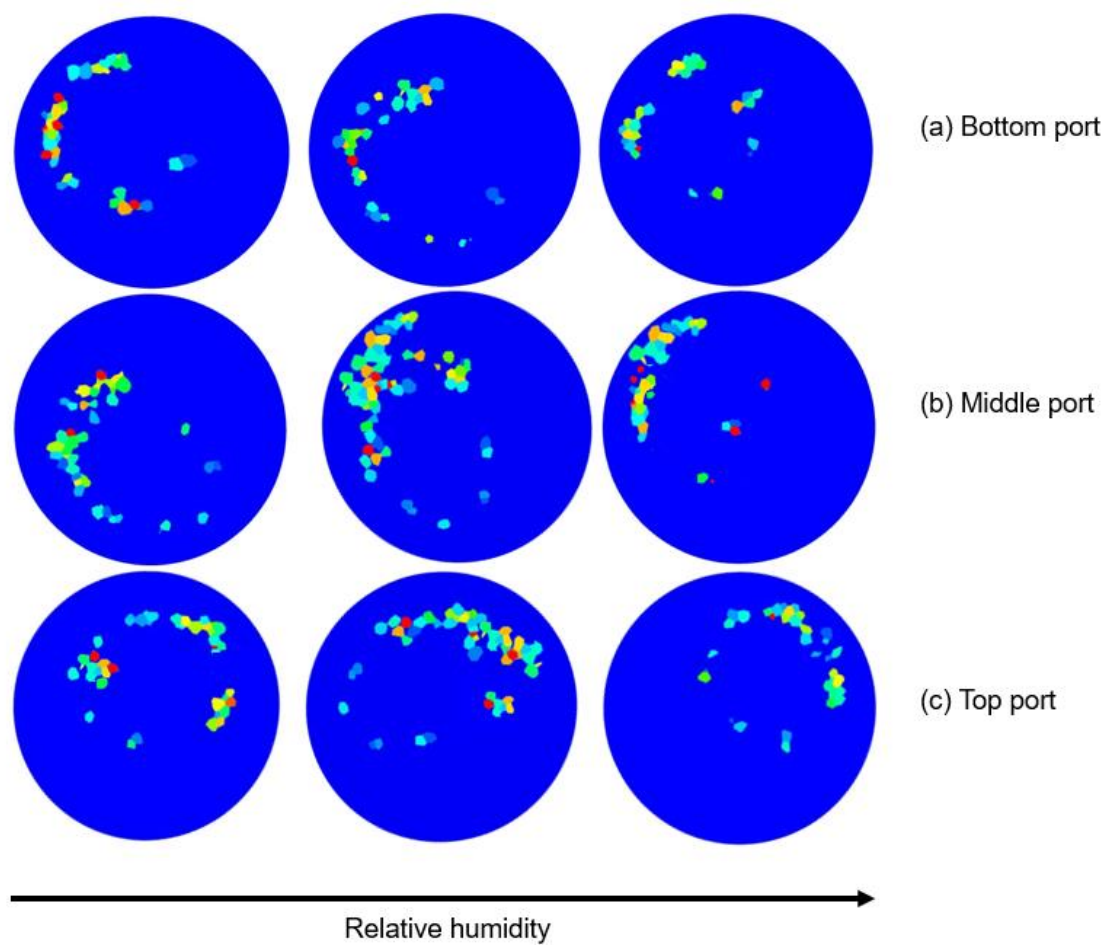


Figure D- 4: Ca(OH)_2 – Modelled radial solids volume fraction at (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas relative humidity

ANNEXURE E POTASSIUM CARBONATE ADDITIONAL RESULTS

E.1 Experimental radial solids distribution for potassium carbonate

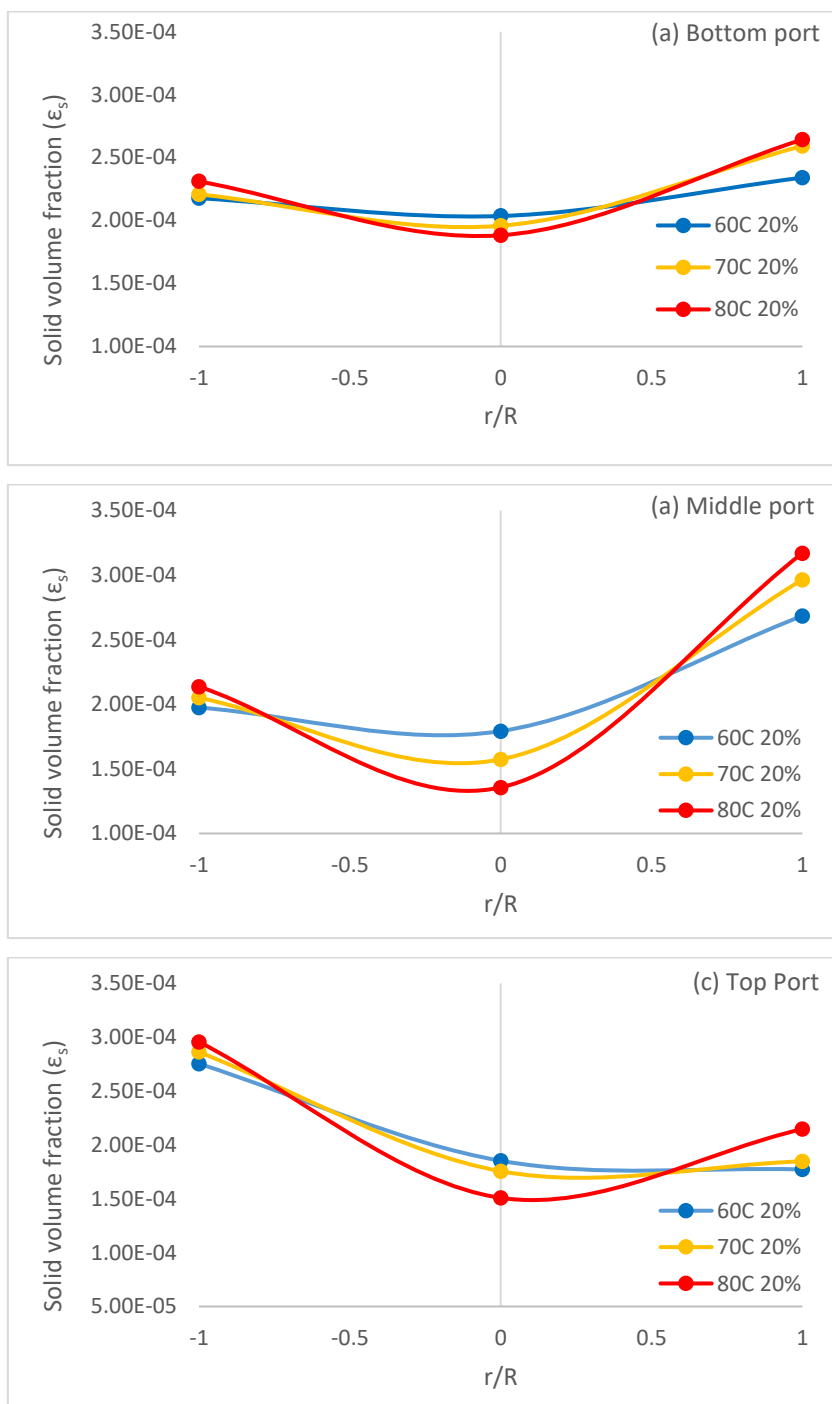


Figure E- 1: K_2CO_3 - Experimental radial solids distribution profile at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas temperature and 20% RH

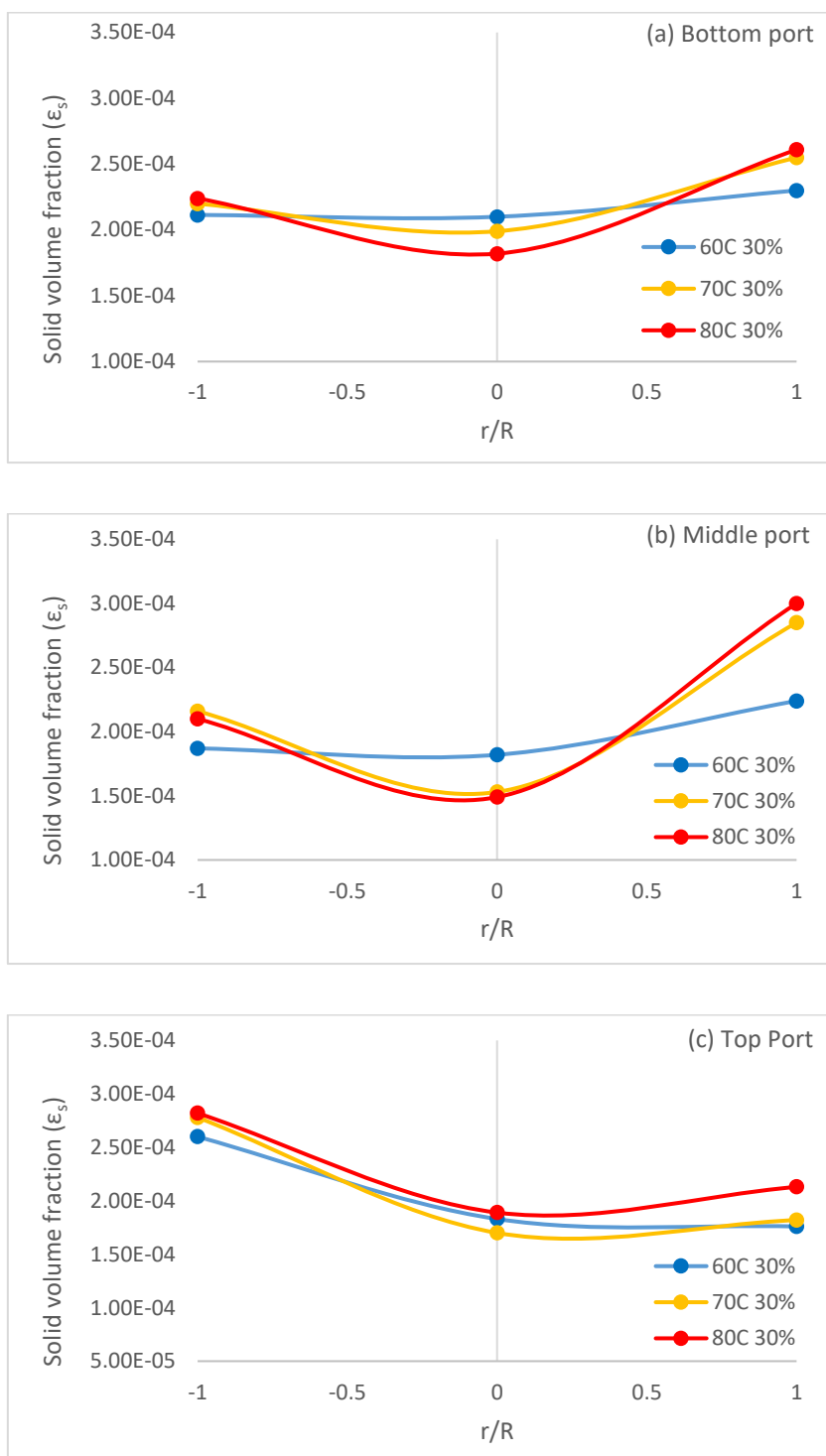


Figure E- 2: K_2CO_3 - Experimental radial solids distribution profile at the (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas temperature and 30% RH

E.2 Modelling radial solids distribution for potassium carbonate

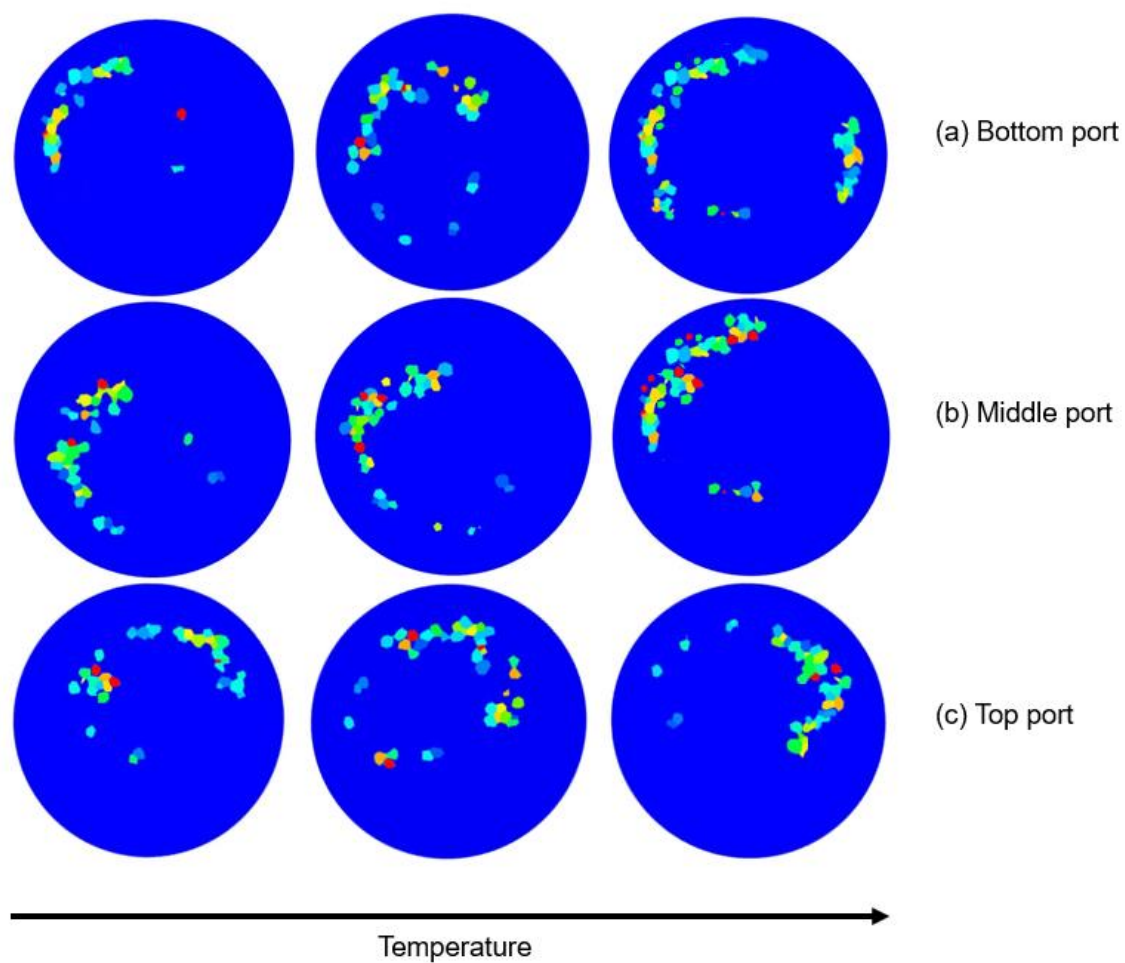


Figure E- 3: K_2CO_3 - Modelled radial solids volume fraction at (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas temperature

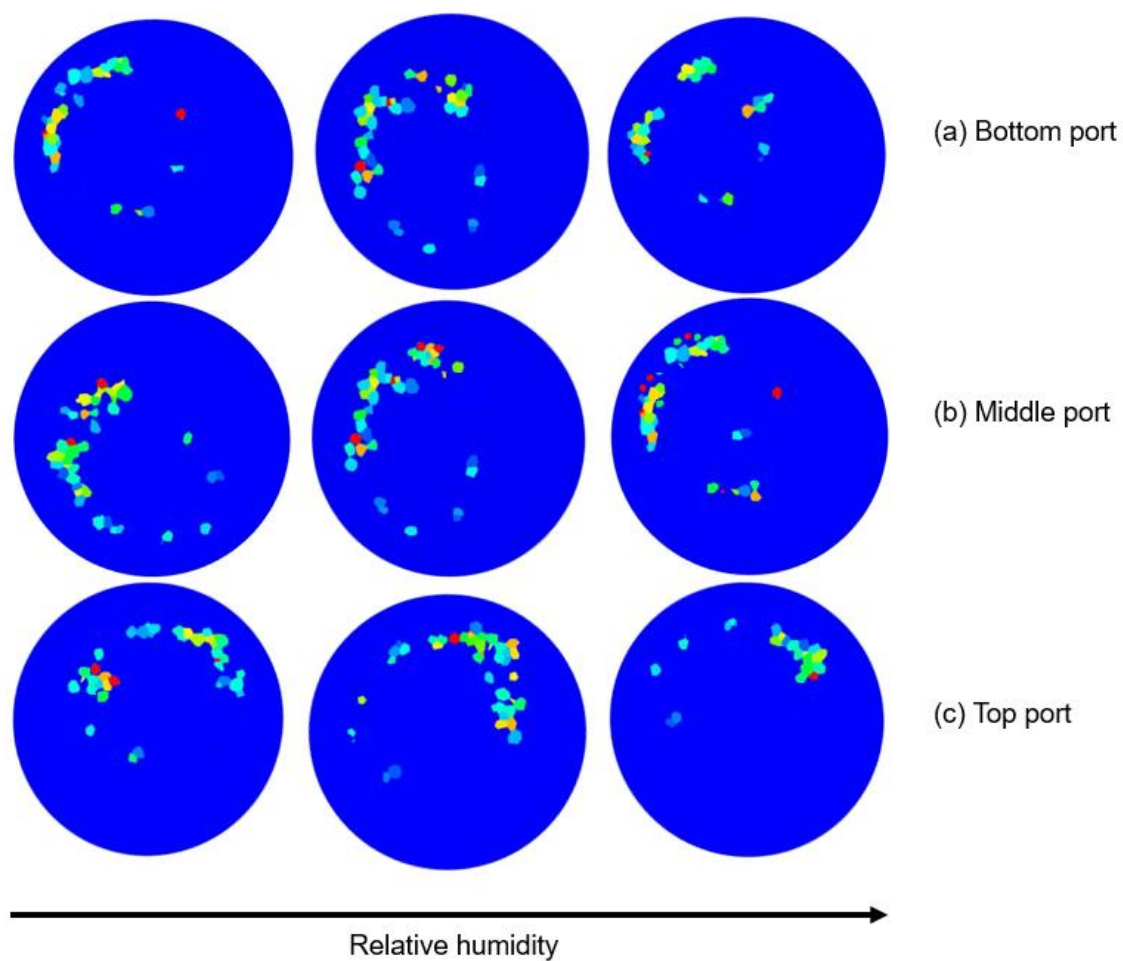


Figure E- 4: K_2CO_3 - K_2CO_3 - Modelled radial solids volume fraction at (a) bottom port, (b) middle port and (c) top port of the riser for varying inlet gas relative humidity