Numerical investigation of gas-solid flow in the calciner



Master's thesis by Kamil Borawski Institute of Energy Technology Thermal Energy and Process Engineering Aalborg University



FLSmidth R&D Centre Dania





Title: Type: Semester: Project period: Supervisors:

Project group:

Numerical investigation of gas-solid flow in the calciner Master's thesis 4^{th} semester of TEPE 1^{st} of February 2009 - 3^{rd} of June 2009 Thomas Joseph Roch Condra Matthias Mandø TEPE4-1045

Copies:5Pages:64Appendices:65 - 84Supplements:On enclosed CD

ABSTRACT:

This report deals with a numerical investigation of gas-solid flow. Simulations are based on a CFD modeling of the flow of raw meal particles in the calciner in FLUENT environment. Results from the simulations were compared with experiments carried out for a model of the calciner placed in FLSmidth R&D Centre Dania. Therefore, the numerical model was build in a way ensuring geometrical similitude with kinematically similar boundary conditions to the calciner used in the experiments.

A description of building the model involves several aspects *e.g.* a mesh generation and grid-independence study, an analysis of a sufficient number of particles tracked in the simulations and relevant calculations required for determination of boundary conditions. Two cases were considered in present work: the flow with strong and weak swirling effect. Thereby, the influence of revolving flow on a distribution pattern of raw meal particles was investigated. Finally, the results from the simulations were compared and analyzed with the experimental data.

Preface

This report has been written under the *Thermal Energy and Process Engineering* graduate programme at the *Institute of Energy Technology* - AAU. This project involves a numerical investigation of gas-solid flow and it was done in a very close co-operation with FLSmidth A/S - one of the world's largest suppliers of process equipment to the cement and mineral industry.

The project work is mainly focused on the numerical simulations of gas-solid flow in the calciner. The simulations have been investigated in order to validate experiments described in Borawski (2008). The test devices have been built at FLSmidth's Research and Development Centre Dania. The simulations were carried out in FLUENT version 6.3.26 (an academic licence). A grid of the model were made in Gambit version 2.3.16 (an academic licence).

The report consists of three parts: the main report, a set of appendices and a CD, where electronic version of the report as well as all of case files are placed. The report can be read independently of the appendices and supplements, but is substantiated by these. Tables and figures have been enumerated with the number of the chapter and the number of the figure in that chapter, *e.g.*"Figure 3.1". This figure will be the first figure in Chapter 3. Appendices are indicated with letters, *e.g.*"Appendix A". Citations have been made in the Harvard method - (Surname, year).

Extensive thanks are expressed to all those, who were involved in this project and have provided any help in solving encountered problems. Among others, to Ejnar Jessen who created a possibility of doing this project, to Michael Bo Dragsdal Hansen who supervised the project and offered a lot of suggestions, to Morten Drivsholm for his innumerable ideas, to Ágúst Ørn Einarsson and Søren Hundebøl for their help in many aspects related to the project, and to all FLSmidth's employees for their kindness.

Kamil Borawski Aalborg, June 2009

Summary

This report concerns a numerical investigation of gas-solid flow. Simulations are based on a CFD modeling of the flow of raw meal particles in the calciner in FLUENT environment. The main objective of the simulations was to determine a dispersion pattern of raw meal particles within the calciner for two types of the flow. The results from the simulations were compared and analyzed with experiments carried out by Borawski (2008). The experiments were based on temperature measurements in various cross-sections of a model of the calciner placed in FLSmidth R&D Centre Dania. Based on measured temperature, a distribution pattern of the particles in the calciner was determined. Therefore, a CFD model was build in a way ensuring geometrical similitude with kinematically similar boundary conditions to the calciner used during the experiments.

The report includes a description of a system used during the experimental work. Next, relevant parameters describing the flow of raw meal particles in the calciner were determined according to a theoretical background introduced in this report. Based on these informations, an appropriate turbulence and multiphase model was chosen, namely the $k - \varepsilon$ model and the DPM model, respectively. Next step was to build a mesh of the calciner. The mesh was created in GAMBIT. A grid-independence study were carried out in order to determine an appropriate number of cells. Thus, the mesh used in the simulations contains approx. 350,000 cells. A statistically independent solution for the simulations including a dispersed phase was found for 24,000 particles tracked. In order to determine an appropriate boundary conditions corresponding to the experiments, relevant calculations were investigated.

The simulations were carried out for two cases. Case 1 is the flow with strong swirling effect, whereas case 2 with weak. The results from the simulations for both cases were compared regarding flow features and dispersion pattern of raw meal particles within the calciner. Thereby, the influence of revolving flow on a distribution of the particles was investigated. Finally, the results from the simulations were compared and analyzed with the experimental data. Two methods were used for this comparison. The first method is based on plotting the temperatures measured in the experiments and corresponding temperatures resolved from the simulations. The second method is based on plotting the so called weight factor in form of contour plots. The weight fraction corresponds to a fraction of raw meal particles in a certain area of the calciner. At the end of this report, final conclusions are drawn and proposals for the future work are stated.

Nomenclature

| Symbol | Description | (Unit) |
|--------------|--|----------------------|
| Ap | Projected particle area | (m ²) |
| Bi | Biot number | (-) |
| C, C_{μ} | Dimensionless constants | (-) |
| Cp_a | Heat capacity of air | (J/kg·K) |
| C_c | Cunningham correction factor | (-) |
| Cp_s | Heat capacity of raw meal | (J/kg·K) |
| C_D | Drag coefficient | (-) |
| d_p | Particle diameter | (m) |
| D | Diameter of calciner | (m) |
| f | Drag modification factor | (-) |
| f_c | Collision frequency | (-) |
| $ec{F}$ | Force vector | (N) |
| F_D | Drag force | (N) |
| Fr | Froude number | (-) |
| ġ | Gravity vector | (m/s^2) |
| h | Heat transfer coefficient | (W/m²·K) |
| h_d | Height of down part of spreader box | (m) |
| h_u | Height of upper part of spreader box | (m) |
| Ι | Turbulence intensity | (-) |
| k | Turbulent kinetic energy | (m^2/s^2) |
| k_a | Thermal conductivity of air | (W/ _{m·K}) |
| k_s | Thermal conductivity of raw meal | (W/ _{m·K}) |
| l | Characteristic length scale | (-) |
| L | Distance between particles | (m) |
| 'n | Mass flow of raw meal | (kg/s) |
| 'n | Mass flow of air | (kg/s) |
| m_p | Mass of particle | (kg) |
| Ma | Mach number | (-) |
| n | Number density | (-) |
| n | Spread parameter in Rosin-Rammler distribution | (-) |
| n_t | Number of particles | (particles/s) |
| N | Number of cells in a cross-section | (-) |
| Nu | Nusselt number | (-) |
| р | Pressure | (Pa) |
| Pr | Prandtl number | (-) |

| 0 | Heat transferred to particles | (\mathbf{W}) |
|----------------------|--|------------------------|
| Q_s | Batio of tortiary air to total flow | (\mathbf{v}) |
| r_t | Particle Reynolds number | (-) |
| ne _p | Surface of particle | (-) |
| Sp St | Surface of particle | (111) |
| 51 | | (-) |
| <i>l</i> <i>→</i> | | (S) |
| u | Fluid velocity vector | (III/s) |
| <u>u</u> | Velocity component in x direction | (m/s) |
| u _. | Mean velocity of fluid | (m/s) |
| <i>u'</i> | Fluctuating velocity of fluid | (m/s) |
| $u_{	au}$ | Friction velocity | (m/s) |
| u_{la} | Velocity of leakage air | (m/s) |
| u_s | Speed of sound | (m/s) |
| U | Mean velocity in calciner | (m/s) |
| \vec{v} | Particle velocity | (m/s) |
| ν | Velocity component in <i>y</i> direction | (m/s) |
| $\dot{V_{la}}$ | Flow of leakage air | (m^{3}/s) |
| V_{p} | Volume of particle | (m ³) |
| V_{rm} | Volumemetric flow of raw meal | (m^3/s) |
| V_{rc} | Volume of rotary chamber | (m^3) |
| , r V | Total airflow | (m^{3}/s) |
| T_{a} | Air temperature | (K) |
| T _u | Raw meal temperature | (K) |
| T | Wall temperature | (K) |
| | Mass fraction | (-) |
| 1a 10 | Velocity component in z direction | (m/s) |
| <i>u</i> 7 | Mass loading | (|
| | Mass foading | (-) |
| Greek sym | bols | |
| α | Angle of adjustable plate in the SB | (°) |
| α_d | Volume fraction of raw meal | (-) |
| δ_{ii} | Kronecker's delta function | (-) |
| ε | Turbulent dissipation | (m^2/s^3) |
| ϕ | Sphericity | (-) |
| $\dot{\eta}$ | Kolmogorov length scale | (m) |
| $\dot{\lambda}$ | Molecular mean free path | (m) |
| ν | Kinematic viscosity | (m^2/s) |
| Vτ | Turbulent kinematic viscosity | (m^2/s) |
| μ | Dynamic viscosity | $(Pa \cdot s)$ |
| | Turbulent dynamic viscosity | $(Pa \cdot s)$ |
| ም 1 ብ | Characteristic velocity scale | (-) |
| 0 | Fluid density | (kg/m^3) |
| р 01. | Raw meal hulk density | (kg/m^3) |
| P US | Raw meal density | (kg/m^3) |
| μs σ | Normal strasses | (N/m^2) |
| τ | Shoar strassos | (N/m^2) |
| ι Τ | Valmagaray time scale | (**/III ⁻) |
| ι - | Rollingorov unie scale | (8) |
| ι_L | Euroy unite scale | (S) |

| $	au_p$ | Relaxation time | (s) |
|----------|---|-----|
| τ_s | Characteristic time scale | (s) |
| $	au_c$ | Time between particle-particle collisions | (s) |

Abbreviations

| AR - Aspect | ratio |
|-------------|-------|
|-------------|-------|

- BBO Basset-Boussinesq-Oseen
- CFD Computational Fluid Dynamic
- CTE Crossing Trajectory Effect
- CPU Central Processing Unit

DNS - Direct Numerical Simulations

DPM - Discrete Phase Model

DRWM - Discrete Random Walk Model

EAS - EquiAngle Skew

ILC - In-Line Calciner

LES - Large Eddy Simulations

NS - Navier-Stokes

PDF - Probability Density Function

PSD - Particle Size Distribution

RANS - Reynolds-averaged Navier-Stokes

RMS - Root Mean Square

RSM - Reynolds-stress Model

SB - Spreader Box

SC - Size change

TA - Tertiary Air

List of Figures

| 1.1 | Typical ILC kiln system | 3 |
|------|--|----|
| 1.2 | The model of the ILC calciner located in FLSmidth R&D Centre Dania | 4 |
| 1.3 | The characteristic dimensions of the model of the ILC calciner located in FLSmidth R&D Centre Dania. | 6 |
| 2.1 | The drag curve of a rigid sphere for steady-state conditions. | 14 |
| 2.2 | Two-phase coupling regions for particle-fluid turbulence interaction | 17 |
| 2.3 | Schematic comparison of the particle trajectories for different Stokes numbers. | 18 |
| 2.4 | Schematic of the interaction between particles and turbulent eddies | 21 |
| 3.1 | Domain subdivided on five subdomains. | 26 |
| 3.2 | The y^+ values along the calciner. | 27 |
| 3.3 | Regions of the bad-quality cells of the meshed calciner | 28 |
| 3.4 | Final mesh of the calciner | 29 |
| 3.5 | Residuals of the simulations carried out for the coarse grid. | 30 |
| 3.6 | Comparison of the velocity profiles for grids with different number of cells | 31 |
| 3.7 | Microscope view of the raw meal particles | 33 |
| 3.8 | Drag coefficient for non-sherical particles with various sphericities | 34 |
| 3.9 | Particle size distribution (Rosin-Rammler type) | 35 |
| 3.10 | The feeding system of raw meal | 36 |
| 3.11 | Raw meal injection into the calciner. | 36 |
| 3.12 | The surface of release the raw meal particles defined in FLUENT. | 37 |
| 3.13 | The air leakage through the rotary valve | 37 |
| 3.14 | Progress of residuals for simulations carried out for the flow with raw meal particles | 39 |
| 3.15 | Comparison of a particle concentration for different number of particles | 42 |
| 4.1 | The velocity magnitude displayed on transverse contours for two types of the flow. \ldots | 46 |
| 4.2 | The velocity vectors at the bottom of the calciner. | 47 |

LIST OF FIGURES

| 4.3 The velocity vectors of the lower cylindrical part of the calciner. | 47 |
|---|--|
| 4.4 The velocity vectors of the upper cylindrical part of the calciner. | 48 |
| 4.5 Raw meal distribution displayed on contour plots for both types of the flow. | 50 |
| 4.6 The residence time of the raw meal particles in the calciner. | 51 |
| 4.7 The rod with thermocouples in the calciner. | 52 |
| 4.8 Coordinates of thermocouples placed in the calciner during the experiments. | 53 |
| 4.9 Relation between temperature and raw meal concentration. | 53 |
| 4.10 Comparison of the temperature profiles for the experiments and the simulations. | 54 |
| 4.11 Comparison of the temperature profiles for the experiments and the simulations. | 54 |
| 4.12 Comparison of the temperature profiles for the experiments and the simulations. | 55 |
| 4.13 Comparison of the temperature profiles for the experiments and the simulations. | 55 |
| 4.14 The contour plots of the weight factor for both types of the flow. | 57 |
| 4.15 The contour plots of the weight factor for both types of the flow. | 57 |
| 4.16 The contour plots of the weight factor for both types of the flow. | 58 |
| 4.17 The contour plots of the weight factor for both types of the flow. | 58 |
| A.1 Cement production | 67 |
| | |
| D.1 Comparison of the velocity profiles for grids with different number of cells | 75 |
| D.1 Comparison of the velocity profiles for grids with different number of cells | 75 75 |
| D.1 Comparison of the velocity profiles for grids with different number of cells | 75 75 76 |
| D.1 Comparison of the velocity profiles for grids with different number of cells D.2 Comparison of the velocity profiles for grids with different number of cells D.3 Comparison of the velocity profiles for grids with different number of cells E.1 Particle size distribution of raw meal before the experiments | 75 75 76 78 |
| D.1 Comparison of the velocity profiles for grids with different number of cells | 75 75 76 78 79 |
| D.1 Comparison of the velocity profiles for grids with different number of cells | 75 75 76 78 79 81 |
| D.1 Comparison of the velocity profiles for grids with different number of cells | 75 75 76 78 79 81 81 |
| D.1 Comparison of the velocity profiles for grids with different number of cells | 75 75 76 78 79 81 81 82 |
| D.1 Comparison of the velocity profiles for grids with different number of cells | 75 75 76 78 79 81 81 82 82 |
| D.1 Comparison of the velocity profiles for grids with different number of cells | 75 75 76 78 79 81 81 82 82 82 |
| D.1 Comparison of the velocity profiles for grids with different number of cells | 75 75 76 78 79 81 81 82 82 82 82 83 |
| D.1 Comparison of the velocity profiles for grids with different number of cells | 75 75 76 78 79 81 81 82 82 82 83 83 |
| D.1 Comparison of the velocity profiles for grids with different number of cells | 75 75 76 78 79 81 81 82 82 82 82 83 83 84 |
| D.1 Comparison of the velocity profiles for grids with different number of cells | 75 75 76 78 79 81 81 82 82 82 82 83 84 84 |
| D.1 Comparison of the velocity profiles for grids with different number of cells | 75 75 76 78 79 81 81 82 82 82 83 83 84 85 |

| G.6 Comparison of the temperature profiles for the experiments and the simulations | 85 |
|---|----|
| G.7 Comparison of the temperature profiles for the experiments and the simulations. \ldots | 85 |
| G.8 Comparison of the temperature profiles for the experiments and the simulations | 86 |
| G.9 Comparison of the temperature profiles for the experiments and the simulations | 86 |
| G.10 Comparison of the temperature profiles for the experiments and the simulations. \ldots | 86 |
| G.11 Comparison of the temperature profiles for the experiments and the simulations. | 87 |
| G.12 Comparison of the temperature profiles for the experiments and the simulations. | 87 |

List of Tables

| 1.1 | The characteristic process parameters of the system. | 5 |
|-----|--|----|
| 2.1 | Regimes of the flow for raw meal particles moving in the calciner. | 16 |
| 2.2 | The relaxation time and the Stokes number for raw meal particles moving in the calciner. \ldots . | 18 |
| 2.3 | The time ratio of the relaxation time of raw meal particles to the time between collisions. \ldots . | 19 |
| | | |
| 3.1 | Example of the quality types of the cells. | 28 |
| 3.2 | Typical proportion of the materials composing a raw mixture (Alsop, 2005) | 31 |
| 3.3 | Chemical composition of the raw mixture on ignited basis (Alsop, 2005). | 32 |
| 3.4 | Parameters describing the Rosin-Rammler distribution. | 35 |
| 3.5 | The inlet conditions of raw meal injection defined in FLUENT | 38 |
| 3.6 | Optional forces and effects affecting particles | 41 |
| 3.7 | The thermal response time for raw meal particles moving in the calciner. | 44 |

Contents

| 1 | Intr | oduction | 1 |
|---|------|---|----|
| | 1.1 | Environmental aspects | 1 |
| | 1.2 | Calciner | 2 |
| | 1.3 | Problem formulation | 6 |
| | 1.4 | Outline of the report | 7 |
| 2 | Gas | -solid systems in turbulent flow | 9 |
| | 2.1 | Modeling of the fluid phase | 9 |
| | 2.2 | Modeling of the particulate phase | 11 |
| | 2.3 | Turbulent dispersion in gas-solid flows | 16 |
| | 2.4 | Summary | 23 |
| 3 | Мос | leling | 25 |
| | 3.1 | Grid design | 25 |
| | 3.2 | Raw meal properties | 31 |
| | 3.3 | Feeding of raw meal | 35 |
| | 3.4 | Settings in FLUENT | 39 |
| | 3.5 | Summary | 44 |
| 4 | Res | ults | 45 |
| | 4.1 | Flow features | 45 |
| | 4.2 | Raw meal distribution | 49 |
| | 4.3 | Residence time | 51 |
| | 4.4 | Experiments | 51 |

CONTENTS

| | 4.5 Relation between temperature and concentration | 53 |
|----|--|----|
| | 4.6 Simulations vs experiments | 54 |
| | 4.7 Summary | 59 |
| 5 | Conclusion and discusion | 61 |
| | 5.1 Future work | 62 |
| Bi | bliography | 62 |
| A | Cement production | 65 |
| B | Scaling gas-solid flows | 69 |
| С | The Navier-Stokes equations | 71 |
| | C.1 How to solve Navier-Stokes equations? | 72 |
| D | Comparison of velocity profiles for grids with various number of cells | 75 |
| E | Particle size distribution | 77 |
| F | Particle concentration profiles for simulations with various number of particles | 81 |
| G | Comparison of temperatures profiles for simulations and experiments | 83 |

1

Introduction

1.1 Environmental aspects

In recent years, big effort is put on minimize emissions of carbon dioxide and energy consumption. There are two main reasons of this fact: the global warming (CO_2 is a primary greenhouse gas responsible for the greenhouse effect) and depletion of the crude oil reserves (energy consumption simply means fuel consumption). In many cases, these two aspects are dependent of each other, because less fuel consumption usually means less carbon dioxide emission.

The European Union has made a commitment, according to the Kyoto Protocol, that each EU member (with some exceptions) needs to reduce CO_2 emitted to the atmosphere by 8% by 2012 based on the emission level in 1990, regardless of the increasing energy consumption (United Nations, 2007). For this reason, measures have to be taken to limit the global warming. Two possible approaches are described below according to Maheshwari (2009):

• Cap and trade method

A central authority (usually a government) sets a cap on the amount of carbon dioxide that can be emitted. Thus, companies are obligated to hold the level of emission permits (which is equivalent to a certain number of credits). The total number of credits cannot exceed the limit resulting from the Kyoto Protocol. It means, that companies that pollute more than it is allowed, have to buy credits from those who pollute less.

Carbon tax

A carbon tax is an environmental tax on emissions of carbon dioxide and other greenhouse gases. It intends to discourage polutting by charging the tax for high-carbon emissions products.

To sum up, excepting fact that reduction of carbon dioxide is obligatory, to some extent can be also profitable. Furthermore, factories which use greener technology can be more attractive to the customers.

1.1.1 CO₂ in the cement industry

Problem of CO_2 emission to a large degree concerns cement plants (the description of a typical cement plant can be found in Appendix A). Production of one metric ton of a cement re-

1 Introduction

sults in the emission of roughly one metric ton of CO_2 and in some cases even more. Around 60% of this amount (on weight basis) is released during the chemical process alone - so called calcination (formula A.1), whilst 35% comes from a combustion of a fuel in a kiln and a calciner. According to Maheshwari (2009), the greenest technologies applied in the cement industry can reduce carbon dioxide emissions by only about 20%. Despite that fact, taking into account huge amount of cement being consumed¹ and the fact, that cement production is continuosly increasing², any reduction in CO_2 emission is worthwhile.

There are several opportunities to reduce the emission of carbon dioxide and decrease the amount of fossil fuels used. One of them could be replacing the old technologies with the new ones. Best-of-class technologies are 7-10% more efficient than the prevailing (Maheshwari, 2009), but obviously are not attractive financially. Other possibility is to install the waste heat recovery systems as the plants reject large amount of hot gases with the temperature around 300°C. It is also possible to use alternative fuels such as municipal solid waste, waste tyres, industrial wastes, etc. Alternative fuels are very promising as significant amount of fossil fuels could be replaced by using them. Another option could be an oxy-fuel technology, where the air required for combustion processes in the kiln and the calciner would be replaced with the oxygen. Thus, exhaust gases mainly would comprise of steam and carbon dioxide ready for a sequestration³.

Any improvement or change made to the system required detailed knowledge about operation of devices, thus mechanisms and processes occurring in a given system have to be well understood. In many cases (especially in cement industry where machinery is rather of big sizes) analyzing devices is possible only in so called model-scale, what gives rise to a new problem which is "proper" scaling. Usually, a similitude of the system in a full-scale and a model-scale can be achieved, when dimensionless numbers, coming from a dimensional analysis are equal. The dimensional analysis treatment of dilute gas-solid flow is presented in Appendix B.

1.2 Calciner

This project deals with the numerical simulation of the cold model of ILC calciner (In-Line Calciner). This model is located in FLSmidth R&D Centre Dania. The scaling of the model was based on the full-scale ILC calciner placed at Barbetti Cement Plant (Gubbio, Italy) - detailed description of the model can be found in Borawski (2008). A brief description of the operation of the model, as well as process parameters are presented in paragraph 1.2.2.

1.2.1 ILC calciner

The ILC calciner is a part of an ILC kiln system. A typical ILC kiln system is shown on Figure 1.1. A raw mixture is fed to the first top cyclone of the cyclone preheater and in every stage, it

¹In 2007, the United States consumed 110.3 millions metric tons of Portland cement (Portland Cement Association, 2008)

²Currently cement consumption is declining due to the financial crisis, but probably it will return to its previous growth rate.

³Sequestration is a technique used for long-term storage of carbon dioxide.

encounters an exhaust gases coming from the rotary kiln and the calciner (CO_2 from the calcination and smoke gases from the combustion). Preheated raw meal from the second last cyclone is introduced into the calciner, where the calcination occurs. The heat required for calcination process comes from a combustion of a fuel, which takes place at the bottom of the calciner. Calcined raw meal is transported with the exhaust gases to the last bottom cyclone of the preheater, where is separated and finally enters the rotary kiln.



Figure 1.1: Typical ILC kiln system. The picture was made by Borawski (2008).

The ILC calciner is a vessel (in a shape of a pipe), usually comprised of two parts (down and upper) separated with a contraction (see Figure 1.2). One of the reasons of splitting the calciner is to minimize NO_x emission. At the bottom of down part, hot gases coming from the kiln (kiln gases) are introduced. A feed of the fuel for the combustion is placed slightly higher. In the middle of the cone, above the place where the fuel is injected, the tertiary air, preheated in the cooler is introduced and the initial combustion takes place. The way of introducing the tertiary air causes a swirling effect in order to improve the mixing of the fuel and the raw meal. At the beginning of the cylindrical part of the calciner, a part of the raw mixture is fed. The rest of the raw meal is fed right above the converging/diverging section (at the beginning of the upper part). The feed of the raw meal is done through specially designed boxes (so called distribution or spreader boxes). The purpose of using spreader boxes is to obtain better distribution of raw meal, when it enters the calciner. A suspension of smoked gases and raw mixture is transported to the cyclone through the pipe called "swan neck", where the final calcination and combustion take place.

1 Introduction

1.2.2 Model at FLSmidth R&D Centre Dania

The model of the ILC calciner located in FLSmidth R&D Centre Dania is shown on Figure 1.2. The main purpose of building this model was to determine a distribution of raw meal injected to the upper part of the calciner. The model is running with preheated air (up to 80°C) and cold raw meal (room temperature). Therefore, there is no possibility for carrying out the combustion and the calcination. Experiments consist in mapping the temperature in various cross-sections of the calciner (Figure 1.2) - more detailed description can be found in Section 4.4. The analysis of the raw meal distribution is based on the temperature difference between the air and the raw meal.



Figure 1.2: The model of the ILC calciner located in FLSmidth R&D Centre Dania. The picture on the left was taken in February, 2009. The drawing on the right was made by A. Einarsson (FLSmidth's engineer) and modified by the author. The red lines shows the places of the temperature measurements.

Air inlet

The air flow in the calciner is done by using a fan, which is placed after the whole system. This means that there is an underpressure in the system, because the air is sucked into the calciner and blown away from the fan. The air is introduced into the calciner at two places. From the bottom, the air enters the calciner vertically. This inlet imitates the gases coming from the rotary kiln and it is called the kiln gases. The second inlet is placed in the conical part of the calciner, where the air is blown causing a swirling of the flow. This inlet imitates the tertiary air coming from the cooler. Figure 1.2 presents the model of the calciner, where both the kiln gases and the teriary air are marked.

Feed of raw meal

The raw meal is fed into the calciner right after the restriction between down and upper part. The raw meal is transported through the pipe coming from the feeder, which controls the feeding rate. There is an airlock rotary valve between the feeder and the calciner due to the safety reasons, but also it ensures that an additional air does not enter the system (due to the underpressure in the system). However, a leakage of the air was noticed during the experiments and further analysis of this phenomena can be found in Section 3.3.2. The raw meal coming from the rotary valve enters the Spreader Box (SB) and then the calciner. The main component of the spreader box is a plate with an adjustable angle, which gives the possibility of change the direction of the injected raw meal. In Section 3.3, more detailed description concerned the spreader box and raw meal injection are presented.

Dimensions and process parameters

The most important dimensions of the calciner used for mesh construction and FLUENT settings are shown on Figure 1.3. The process parameters are exactly the same like those set during the experiments carried out by Borawski (2008). It has to be mentioned that several settings have been examined. However, present report concerns only two of them (different ratio of the tertiary air to the total flow in the calciner). The process parameters required for further analysis are shown in Table 1.1. The properties of the raw meal used in the experiments are described in Section 3.2.

| Symbol | Description | Value (unit) |
|-----------|--|--------------------------|
| ṁ | The feeding rate of the raw meal | 302 (kg/h) |
| \dot{V} | The total air flow | 0.96 (m ³ /s) |
| T_a | The temperature of the air | 80°C |
| T_{rm} | The temperature of the raw meal | 27°C |
| α | The angle between the adjustable plate | 30° |
| | in the SB and the horizontal plane | |
| r_t | The ratio of the tertiary air | 20%, 55% |
| | to the total flow | |

Table 1.1: The characteristic process parameters of the system.

1 Introduction



Figure 1.3: The characteristic dimensions of the model of the ILC calciner located in FLSmidth R&D Centre Dania. The drawing was made by A. Einarsson (FLSmidth's engineer) and modified by the author.

1.3 Problem formulation

This project aims to validate an experimental work done by Borawski (2008). The experiments concerned a gas-solid flow in a model of calciner placed in FLSmidth R&D Centre Dania. The calciner is specifically the process equipment that is very important in a cement production. The main aspect brought up regarding the calciner is a distribution of raw meal particles. It is very important that particles are well distributed (an uniform distribution is desirable) for the sake of better efficiency of the cement production. Therefore, this thesis concerns an investigation of the particle distribution in the calciner.

The validation of the experimental results was performed by modeling the flow of raw meal particles in the calciner. Thereby, the main challenge of this project was to build a reliable CFD model based on the calciner used in the experiments and compare relevant data regarding the raw meal distribution.

The process of building the model involved a mesh generation with proper and simultaneously efficient number of cells, an analysis of a sufficient number of particles tracked in simulations and a selection of boundary conditions similar to these in the experiments. Results obtained from the simulations and the experiments were compared graphically and an appropriate conclusions were drawn. **1.4 Outline of the report**

1 Introduction

2

Gas-solid systems in turbulent flow

Generally, gas-solid flow is the phenomena of the transport of particles, which are distinguishable from the carrier phase. Modeling of these systems is very complex because the flow field of the continuous phase¹, as well as motion of the particulate phase, need to be solved. There are many variables which complicate description of gas-solid flows. Therefore, an overview of these systems and the methods used in this project for describing them are presented briefly in this chapter.

Generally, there are two approaches in analyzing fluid mechanics problems:

- *Eulerian method* this method is based on using stationary reference frame. All information about the flow (or particles) are obtained at fixed points in space as fluid (particles) pass through.
- *Lagrangian method* each individual particle (or cloud of particles) is followed as it moves through the domain and its properties are identified as a function of time.

Modeling of the gas-solid systems is also classified based on the type of the reference frame. Again two approaches can be distinguished: the *Eulerian-Eulerian* and the *Eulerian-Lagrangian*. In both cases, the fluid is treated by using an Eulerian reference frame, described below. In Eulerian-Eulerian models, the particulate phase is usually treated as a continuous phase mixed with the fluid phase (these models are also known as *continuum models* or *two-fluid models*). However, these models are popular when the particle loading is high (Shirolkar et al., 1996), thus are not considered in this report (regarding loading see page 19). The Lagrangian particle dispersion models treat particles as discrete objects, and their motion is tracked as they move through the flow field. Usually, only representative samples of the particles are tracked in order to reduce the computational time. However, the number of calculated trajectories should be sufficient to provide a complete picture of the particle behaviour in the turbulent flow (Section 3.4.3).

2.1 Modeling of the fluid phase

One of the most used methods to model the turbulence flow is solving the RANS equations (Reynolds-averaged Navier-Stokes) comprised of the continuity equation Equation 2.2 and

¹Usually, gas is considered as a continuous phase, because the mean free path of the gas molecules is significantly lower than the characteristic size of the particles.

the momentum equation 2.3.² RANS equations arise when the Reynolds decomposition (Equation 2.1) is implemented into the Navier-Stokes equations (Equations C.5). Reynolds decomposition refers to the separation of the flow variable (e.g. velocity) into two components: mean velocity $\overline{u_i}$ and fluctuating velocity u'_i .

$$u_i = \overline{u_i} + u_i' \tag{2.1}$$

After averaging, RANS equations are as follows:

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \tag{2.2}$$

$$\frac{\partial \overline{u_i}}{\partial t} + \overline{u_j} \frac{\partial \overline{u_i}}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \overline{\rho}}{\partial x_i} + \nu \left(\frac{\partial^2 \overline{u_i}}{\partial x_j^2} \right) - \frac{\partial \left(u_i' u_j' \right)}{\partial x_j}$$
(2.3)

where i = 1, 2, 3 stands for direction, , x_i is a distance, $\overline{u_i}$ is a mean velocity, u'_i is a fluctuation velocity, p is a pressure, ρ and v are a density and a kinematic viscosity of the fluid, respectively. It should be noted that equations above are valid for incompressible Newtonian fluid $\rho = \text{const.}$ The averaging of the Navier-Stokes equations gives rise to fluctuating quantities *i.e.* $\rho u'_i u'_j$, called the Reynolds stresses (or the turbulent stresses), which describe the transport of the momentum due to the turbulence (eddies). As a result of appearance of new parameters in RANS equations, there are more unknows than equations - this is so called "closure problem". Within recent years several models have been developed which estimate the missing variables. The $k - \varepsilon$ model used in the simulations concerned this project is described briefly below.

2.1.1 $k - \varepsilon$ model

The $k - \varepsilon$ model is one of the most popular turbulence models used especially in industrial applications (Casey and Wintergerste, 2000). Beside RANS equations, there are two model equations which are solved when $k - \varepsilon$ model is used. These are transport equations for the turbulence kinetic energy k and the rate of dissipation of this energy ε . The $k - \varepsilon$ model is based on the presumption of analogy between the action of viscous stresses and turbulent stresses on the mean flow as it is shown in Equation 2.4 proposed by Boussinesq in 1877 (Versteeg and Malalasekera, 2007). A formula can be written for the viscous stresses according to Equations C.2.

$$-\rho \overline{u_i' u_j'} = \mu_T \left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij}$$
(2.4)

where μ_T is the turbulent viscosity (as distinct from the dynamic viscosity μ , the turbulent viscosity is the feature of the flow, not the fluid), *k* is the turbulent kinetic energy of the flow

²Basically, there are three main methods for solving velocity field of the fluid: RANS, LES (Large Eddy Simulations) and DNS (Direct Numerical Simulations). Usually, LES and DNS are not used in industrial applications due to their high requirements of computer storage and CPU time. A brief description of these methods can be found in Appendix C.1

(Equation 2.5), and δ_{ij} is the Kronecker's delta function ($\delta_{ij} = 1$ if i = j and $\delta_{ij} = 0$ if $i \neq j$) which ensures correctness in the formula for the normal Reynolds stresses.

$$k = \frac{1}{2} \sum \overline{u_i^{\prime 2}} \tag{2.5}$$

Dimensional analysis shows that the turbulent viscosity μ_T is a function of the characteristic velocity scale ϑ and the characteristic length scale l which are representative of the large-scale turbulence. They are defined as follows:

$$\vartheta = k^{1/2} \tag{2.6}$$

$$l = \frac{k^{3/2}}{\varepsilon} \tag{2.7}$$

Hence, the turbulence viscosity can be calculated

$$\mu_T = C\rho \vartheta l = \rho C_\mu \frac{k^2}{\varepsilon} \tag{2.8}$$

where *C* and C_{μ} are dimensionless constants.

The main disadvantage of this model comes from Boussinesq's turbulent viscosity assumption (Equation 2.4) which states that the turbulence of the flow is isotropic - fluctuating components are equal in each direction.

2.2 Modeling of the particulate phase

Models describing particulate phase in the Lagrangian frame are classified for the sake of particle trajectories, into two major groups. The first type of models are based on a Taylor's series approach, which was one of the first attempt of mathematical description of dispersion in turbulent flows (Shirolkar et al., 1996). The particle trajectories are generated directly using a stochastic model, therefore this method does not require solving the Eulerian velocity field (particle are treated as they were fluid particles). In the second type of models, particle trajectories are solved by using the particle equations of motion, and the fluid velocity field is obtained by solving NS equations, e.g. RANS equations - Section 2.1. In this project, model which is based on the particle equation of motion was used.

2.2.1 The particle equation of motion

The trajectory of discrete phase particle can be determined by solving its equation of motion, which results from Newton's Second Law:

$$m_p \frac{d\vec{\mathbf{v}}_i}{dt} = \sum \vec{F}_i \tag{2.9}$$

where m_p is the mass of the particle, \vec{v}_i is the particle velocity and \vec{F}_i represents different forces acting on the particle. Depending on the characteristic of the particles and the continuous phase, forces have various relevance and some of them can be negligible. The equation of motion for single, isolated particle in uniform Stokes flow (see paragraph 2.2.2) which includes most of the forces is presented below. It is so called BBO equation (Basset-Boussinesq-Oseen) (Crowe et al., 1998).

$$m_{p}\frac{d\vec{v}_{i}}{dt} = 3\pi\mu d_{p}\left(\vec{u}_{i} - \vec{v}_{i}\right) + V_{p}\left(-\nabla p + \nabla \cdot \tau\right) + \frac{\rho V_{p}}{2}\left(\dot{u}_{i} - \dot{v}_{i}\right) + \frac{3}{2}d_{p}^{2}\sqrt{\pi\rho\mu}\left[\int_{0}^{t}\frac{\dot{u}_{i} - \dot{v}_{i}}{\sqrt{t - t'}}dt' + \frac{(\vec{u}_{i} - \vec{v}_{i})_{0}}{\sqrt{t}}\right] + m_{p}\vec{g}_{i}$$
(2.10)

where V_d is the volume of the particle, t is the time, t' is the integration time interval, \vec{g}_i is the gravitational acceleration, τ is the shear stress, \dot{u}_i is the material derivative of the fluid velocity $(D\vec{u}_i/Dt)$, \dot{v}_i is the derivative of the particle velocity $(d\vec{v}_i/dt)$, and index 0 represents initial conditions.

In the BBO equation, terms after the equals sign are (in turn from the left):

- **Steady-state drag force** the drag force which acts on the particle in a uniform pressure field (no acceleration of the relative velocity between particle and fluid) this is known as Stokes' law (Rhodes, 2007). Extensive description of this force is presented in the next page.
- **Pressure force** the force exerted on particle immersed in the fluid with noticable pressure gradient (buoyancy if hydrostatic pressure).
- Virtual mass force also called added mass force. This force arises when a particle undergoes acceleration or deceleration, what requires that the fluid also accelerates or decelerates.
- **Basset force** also called the history term. It is arised due to the lagging boundary layer development as the relative velocity changes with time.
- **Body forces** in the BBO equation, only gravity force is included. The other of body forces: the Coulomb force, thermophoretic force, forces which arise when a particle moves in an electric, or magnetic field.

The BBO equation does not include all of the forces. Those not included are listed below:

- **Faxen force** this force is an extension of the steady-state drag force when the flow field is non-uniform.
- Lift forces forces acting on the particle due to particle rotation. Lift forces are clasiffied into three forces (Rosendahl, 1998); the profile lift force is the most common type of lift, which stems from the orientability of the particle; the Saffman lift force is due to nonuniform velocity near solid boundaries what results in nonuniform pressure distribution; the Magnus lift force is due to the particle rotation. This causes a difference in the velocity resulting the difference in the pressure betweeen sides of the particle.

• Torque - is due to the shear stress distribution on the particle surface.

As it was mentioned, forces acting on the particle moving in the fluid can or cannot be neglected, depending on the conditions of the system. Basically, if the fluid-particle density ratio is very small (~ 10^{-3}), the BBO equation can be justifiably simplified to the form presented in Equation 2.11 (Crowe et al., 1998), where all forces are negligible, apart from the drag force and the gravity force. Work, which has been done by Rusås (1998) proves that the drag force was the most dominant force acting on combusting coal particle with a diameter of 30 μ m, accounting for no less than 95% of the total force.

$$m_p \frac{d\vec{v}_i}{dt} = 3\pi \mu d_p \left(\vec{u}_i - \vec{v}_i \right) + m_p \vec{g}_i$$
(2.11)

Usually, in the equation above, the drag force (the first term on the right of the equals sign) is expressed by Equation 2.12 incorporating the drag coefficient C_D .

2.2.2 The steady-state drag force

The steady-state drag force \vec{F}_D is commonly expressed over the entire range of the particle Reynolds number Re_p as:

$$\vec{F}_D = \frac{1}{2} C_D \rho A_p |\vec{u}_i - \vec{v}_i| (\vec{u}_i - \vec{v}_i)$$
(2.12)

where A_p describes the projected particle area. The particle Reynolds number is expressed in the following equation:

$$Re_p = \frac{\rho d_p |\vec{u}_i - \vec{v}_i|}{\mu} \tag{2.13}$$

When combined and rearranged, Equation 2.12, with the term describing Stokes' law in the BBO equation (Equation 2.10) - the first term on the right after the equals sign, the drag coefficient can be expressed as:

$$C_D = \frac{24}{Re_p} \quad \text{or} \quad C_D = \frac{24}{Re_p} f \tag{2.14}$$

where *f* is the modification factor which approaches unity in Stokes' regime. Usually, there are three regimes depending on the particle Reynolds number Re_p : the Stokes regime for $Re_p < 0.1$, the Newton regime for $Re_p > 1000$ (Michaelides, 2005) and the intermediate regime - so called the Allan regime. In the Newton's regime, the flow is considered as fully turbulent and the drag coefficient remains almost constant and attains a value between 0.42 - 0.44 (Michaelides, 2005). There are many empirical and semi-empirical correlations with different accuracy and complexity, which describes modification factor *f* in Allan regime (some of them are extended for all three regimes). The popular ones are listed in Clift et al.

(2005) where is also presented the range of deviation in C_D from the "standard drag curve".³ It has to be mentioned that above certain critical Reynolds number which is $Re_p \approx 3.7 \cdot 10^5$ (Michaelides, 2005), the drag coefficient drastically decreases due to the transition of a boundary layer around particle from laminar to turbulent. The drag curve, which was derived from the correlation proposed by Clift and Gauvin (Equation 2.15) (Clift et al., 2005), is shown on Figure 2.1. This correlation is essentially a correction of the expression by Schiller and Nauman for high Re flows (Michaelides, 2005). The correlation developed by Schiller and Nauman includes only the first term after the equals sign of Equation 2.15 and is valid for Re < 1000.



$$C_D = \frac{24}{Re_p} \left(1 + 0.15Re_p^{-0.687} \right) + \frac{0.42}{1 + 4.25 \cdot 10^4 Re_n^{-1.16}}$$
(2.15)

Figure 2.1: The drag curve of a rigid sphere for steady-state conditions.

Terminal velocity

Considering single particle (sphere) with the density ρ_s falling in the fluid ($\vec{u} = 0$), there are three major forces acting on this particle (the gravity, the buoyancy and the drag force). When the particle reaches its terminal velocity \vec{v}_t , there is no acceleration of the particle, because these forces are in balance ($d\vec{v}/dt = 0$):

$$\frac{1}{2}C_D \rho A_p |\vec{u} - \vec{v}| (\vec{u} - \vec{v}) = V_p (\rho_s - \rho) \vec{g}$$
(2.16)

Using equations describing, the drag coefficient (Equation 2.14), and the particle Reynolds number (Equation 2.13) to solve for the terminal velocity in the equation above, results in

³The standard drag curve is a result of many experimental studies and it presents the drag coefficient of a rigid sphere as a function of the particle Reynolds number for steady-state conditions.

$$\vec{v}_t = \frac{d_p^{\ 2}(\rho_s - \rho)\vec{g}}{18\mu f}$$
(2.17)

Maximum diameter of the particle in the Stokes' regime

The maximum diamater of the particle in the Stokes' regime can be estimated, when the similar procedure to the one above is applied. Basically, this diameter is obtained when the equation for the particle Reynolds number (Equation 2.13) is introduced into Equation 2.17. Hence, the diameter d_{max} can be determined by

$$d_{max} = 1.22 \sqrt[3]{\frac{\mu^2}{(\rho_s - \rho)\rho \vec{g}}}$$
(2.18)

It has to be mentioned, that the maximum diameter estimated by the equation above is valid for $Re_p < 0.1$.

Assuming that the terminal velocity is equal to the slip velocity (difference between the fluid velocity and the particle velocity) for the raw meal particles moving in the calciner, the maximum diameter in the Stokes' regime is $d_{max} \approx 30 \,\mu$ m. It means that bigger particles move in the Allan or the Newton regime. In order to determine the relevant region of the operation, the particle Reynolds number for raw meal particles larger than d_{max} was determined according to the following procedure:

- 1. Determine the particle Reynolds number
- 2. Calculate the terminal velocity from Equation 2.17⁴
- 3. Calculate the drag coefficient from Equation 2.15
- 4. Calculate the drag coefficient from Equation 2.16
- 5. Repeat the procedure until the difference between the drag coefficients calculated in steps 3 and 4 is sufficiently small

Table 2.1 shows the terminal velocity and the particle Reynolds number for smallest, mean (d_{50}) and largest raw meal particles (regarding size of the raw meal particles, see section 3.2.3).

⁴The modification factor f was determined from the correlation proposed by Shiller and Nauman (14).

| Size (µm) | v _t (m/s) | Re_p | Regime |
|-----------|----------------------|---------------------|----------------|
| 1 | $7.5 \cdot 10^{-5}$ | $1.1 \cdot 10^{-4}$ | Stokes' regime |
| 13 | 0.013 | $7.8 \cdot 10^{-3}$ | Stokes' regime |
| 100 | 0.58 | 2.73 | Allan's regime |

Table 2.1: Regimes of the flow for raw meal particles moving in the calciner.

2.3 Turbulent dispersion in gas-solid flows

Turbulent dispersion or turbulent particle dispersion is a commonly used term describing the transport phenomena of particles in a carrier phase, whilst the flow is turbulent. The influence from turbulence on the particle motion is rather significant and has to be considered. However, modeling the motion of the dispersed phase is very complex due to many aspects, which are not very well understood. Nevertheless, a brief introduction to the interaction between phases in gas-solid systems is included in this section. Firstly, the most useful definitions of dispersed phase flows are introduced and determined for system described in this report.

2.3.1 Phase coupling

An important concept in the analysis of gas-solid flows is coupling, which describes an effect of one phase to another. Coupling can take place through momentum transfer (the result of the drag force on phases) and energy transfer (heat transfer between phases). In general, there are three types of coupling:

- **One-way coupling** the particulate phase has a negligible effect on the fluid phase (obviously, the fluid phase affects the particulate phase)
- Two-way coupling a mutual effect between both phases
- Four-way coupling two-way coupling with simultaneous particle-particle interactions (associated only with dense flows)

Several ways of the determination, whether a gas-solid flow under a given conditions is one-, two-, or four-way coupled, can be found in the literature. The results for one of them, proposed by Crowe (2006) are presented on Figure 2.3.1, where a map for coupling interactions depending on the volume fraction (paragraph 2.3.1) and the particle Reynolds number is shown.

Volume fraction

The volume fraction of the dispersed phase is defined as

$$\alpha_d = \frac{V_d}{V} \tag{2.19}$$
where V_d is the volume of dispersed phase in a total volume V. Assuming the uniform distribution of particles in the system, the volume fraction can be estimated as the ratio of the dispersed phase volumetric flow multiplied by an average residence time for particles (approx. 1.5 (s) - see Section 4.3) and the volume of the calciner, where dispersed phase is moving (approx. 1.32 (m³)):

$$\alpha_d = \frac{\frac{0.084 \,\text{kg/s}}{2900 \,\text{kg/m^3}} \cdot 1.5 \,s}{1.32 \,\text{m}^3} \quad \Rightarrow \quad \alpha_d \approx 3.5 \cdot 10^{-5}$$

According to Figure 2.2, the flow of raw meal particles in the calciner is two-way coupled.



Figure 2.2: Two-phase coupling regions for particle-fluid turbulence interaction. The plot is from Crowe (2006).

2.3.2 Relaxation time and Stokes number

The relaxation time (also known as the momentum response time) is the time required for a particle to respond to a change in the surrounding fluid velocity and reach a velocity corresponding to 63% of the fluid velocity. The momentum response time is determined from the equation of motion for a sphere, when only drag forces act on this particle (Equation 2.12) and the particle Reynolds number (Equation 2.13). Combining and rearranging these equations gives rise to the relaxation time, which is expressed by

$$\tau_p = \frac{\rho_s d_p^2}{18\mu} \frac{1}{f}$$
(2.20)

The Stokes number is defined as

$$St = \frac{\tau_p}{\tau_s} \tag{2.21}$$

where τ_s is a characteristic time scale of the system. The characteristic time for the flow through the calciner can be determined as D/U where D is the diameter of the calciner and U is the mean velocity; and it is equal $\tau_s = 0.28$ s. If $St \ll 1$ the particle relaxation time is very low compared to the characteristic time of the system. Thus the particles follow the fluid streamlines closely (the velocities of both, the fluid and the particles are nearly the same). If $St \gg 1$, the particles do not follow changes of the fluid velocity. This behaviour is shown schematically on Figure 2.3.





Table 2.2 shows calculated values for the relaxation time and the Stokes number for smallest, mean and largest raw meal particles moving in the calciner.

| Size (μ m) | τ_p (s) | St |
|-----------------|---------------------|---------------------|
| 1 | $7.7\cdot10^{-6}$ | $2.7 \cdot 10^{-5}$ |
| 13 | $1.3 \cdot 10^{-3}$ | $4.6 \cdot 10^{-3}$ |
| 100 | 0.06 | 0.21 |

Table 2.2: The relaxation time and the Stokes number for raw meal particles moving in the calciner.

2.3.3 Dilute systems

Generally, gas-solids flows can be divided into dilute and dense flows. The motion of particles in dilute systems mainly depends on drag forces, whereas in dense systems it is controlled by particles collisions. Basically, the flow can be considered dilute if the average time between particle-particle collisions τ_c (Equation 2.22) is smaller than the relaxation time τ_p , compared to dense systems where τ_c is bigger than τ_p .

$$\tau_c = \frac{1}{f_c} = \frac{1}{n\pi d_n^2 v_r}$$
(2.22)

Thus, the flow would be considered dilute if

$$\frac{\tau_p}{\tau_c} = \frac{n\pi\rho_s d_p^{-4} v_r}{18\mu} < 1 \tag{2.23}$$

where f_c is the collision frequency, n is the number density, v_r is the relative velocity of the particles with respect to other particles, σ is the standard deviation of the particle fluctuation velocity, and Z is the loading. Further analysis gives rise to the equation describing the maximum diameter of particles in dilute flows for a given conditions and it is expressed as follows (Crowe et al., 1998):

$$d_{dil} < \frac{1.33\mu}{Z\rho\sigma} \tag{2.24}$$

The loading is defined as the ratio of the overall mass flow rate of the dispersed phase flow \dot{m}_d to the overall mass flow rate of the continuous phase \dot{m}_c :

$$Z = \frac{\dot{m}_d}{\dot{m}_c} \tag{2.25}$$

The loading calculated for raw meal injected into the calciner is $Z \approx 0.09$.

The standard deviation of the particle fluctuation velocity σ can be identified as the Root Mean Square (RMS) of the turbulence fluctuations of the carrier phase expressed as $\sqrt{(u')^2}$. The RMS value can be determined from the formula describing the level of the turbulence (the turbulence intensity) which is expressed as

$$I = \frac{\sqrt{(u')^2}}{\overline{u}} \tag{2.26}$$

According to (FLUENT, 2006), the turbulence intensities greater than 10% are considered high and this value was used in further analysis. Thus, for the mean velocity in the calciner equals U = 2.5 m/s, the RMS is equal 0.25. Thus, the diameter of raw meal particles in dilute flows should be smaller than $d_{dil} \approx 1 \text{ mm}$. The time ratio τ_p/τ_c for smalles, mean, and largest raw meal particles moving in the calciner is presented in Table 2.3. From the analysis above, the system described in present report can be treated dilute.

| Size (µm) | Ratio τ_p/τ_c |
|-----------|-----------------------|
| 1 | $1.4 \cdot 10^{-3}$ |
| 13 | 0.018 |
| 100 | 0.14 |

Table 2.3: The time ratio of the relaxation time of raw meal particles to the time between collisions.

Particle spacing

The particle spacing describes the average distance between the dispersed phase elements and it is important to determine if particles can be treated as isolated elements. This parameter can also determine, whether the flow is dilute or dense. The particle spacing is expressed as the ratio of the distance between particles centers *L* and the diamater of these particles d_p (it is assumed that particles are spheres with uniform diameters). It is defined as follows (Crowe et al., 1998):

$$\frac{L}{d_p} = \left(\frac{\pi}{6\alpha_d}\right)^{1/3} \tag{2.27}$$

Thus, the interparticle spacing of raw meal particles in the calciner is

$$\frac{L}{d_p} \sim 25$$

In this case, according to (Crowe et al., 1998), particles could be treated as isolated elements and the influence of the neighbouring particles on the drag and the heat transfer can be neglected. According to Michaelides (2005), when the particle spacing is greater than 3, such mixtures are dilute mixtures. This fact confirmed the results of the calculations of the time ratio, carried out above.

2.3.4 Turbulent particle dispersion

The motion of particles in dilute systems can be well predicted only if certain properties of the carrier phase are known. Therefore, either experiments or numerical simulations have to be investigated. In the models based on the particle trajectories, the most important fluid property is its instantaneous velocity at the particle location. The feature of the turbulence flows is that properties (e.g. velocity) are fluctuating in each direction (Tennekes and Lumley, 1972). This is because turbulent flows contain eddies of various sizes - from the characteristic dimension of the system (e.g. diameter of the calciner) to the smallest eddies at which the turbulence kinetic energy is dissipated (the Kolmogorov scale). In general, the turbulent particle dispersion is because eddies act on particles. This is illustrated on Figure 2.4.

The figure above depicts different size of eddies acting on particles of various sizes. The characteristic size of the eddy and the particle size are important parameters in determining the eddy-particle interaction.

Turbulent length and time scales

As the mentioned above, size of the eddies varies from the characteristic length of the system - so called integral length scale, to the smallest eddies possible at which the energy contained in these eddies is dissipated. The size of the smallest eddies is described by the Kolmogorov length scale η and can be estimated according to Equation 2.28. Also, an important parameter is the lifetime of the eddy - the time for which the eddy maintains its original size before



Figure 2.4: Schematic of the interaction between particles and turbulent eddies.

it breaks down or vanishes due to the energy dissipation. Similarly, the Kolmogorov time scale τ is the time associated with the lifetime of the smallest eddies and it is expressed by Equation 2.29.

$$\eta = \left(\frac{v_T^{3}}{\varepsilon}\right)^{1/4} \tag{2.28}$$

$$\tau = \left(\frac{\nu_T}{\varepsilon}\right)^{1/2} \tag{2.29}$$

where v_T is the turbulent kinematic viscosity and ε is its dissipation rate. The turbulence viscosity for the flow in the calciner ($r_t = 20\%$; the rest of parameters according to Table 1.1) is in the range from ~ 10^{-3} to $6 \cdot 10^{-2}$, where higher values relate to the core of the flow. The dissipation rate varies depending on the structure of the calciner - values are significantly higher for the conical part, the converting/diverting section between down and upper part and the swan neck. The turbulence characteristic length and time in the highly turbulent regions (e.g. the swan neck) for $v_T = 0.01$ and $\varepsilon = 300$ are as follows:

$$\eta = 7.6 \cdot 10^{-3} \text{ (m)}$$
 $\tau = 5.8 \cdot 10^{-3} \text{ (s)}$

In gas-solid flows, 3 types of particles can be distinguished (Shirolkar et al., 1996):

- **small particles** if the characteristic dimension is smaller than the Kolmogorov length scale
- **medium particles** if the characteristic dimension is between the Kolmogorov length scale and the integral length scale
- large particles for diameters comparable to the integral scale

The Kolmogorov length scale calculated above is much smaller than the size of raw meal particles. Thus, the particles can be treated as small according to the definition above. The

Kolomogorov time scale is comparable (or even smaller) to the relaxation time of biggest particles (see Table 2.2). It means that these particles do not follow the streamline of the smallest eddies.

Eddy lifetime model

Small particles when introduced to a turbulent flows are "trapped" inside an eddy and their trajectories are associated with the eddy motion. Each eddy is characterized by a velocity (fluctuating component), a size (length scale) and a lifetime (also known as the fluid time scale) and these properties need to be determined. It is assumed that the fluid velocity associated with a particlular eddy is constant within the lifetime and the size of the eddy (or more specifically within the eddy-particle interaction time). A general expression for the eddy length scale l_e and the time scale τ_L is based on local turbulent properties (k and ε) and it is written as

$$\tau_L = A \frac{k}{\varepsilon} \tag{2.30}$$

$$l_e = B \frac{k^{3/2}}{\varepsilon} \tag{2.31}$$

where *A* and *B* are experimentally determined constants, which can be found in the literature (e.g. (Shirolkar et al., 1996)). It has to be mentioned that equations above are determined assuming that the flow is isotropic. The fluid fluctuating velocity is randomly determined from a Probability Density Function (PDF) obtained from local turbulence properties. The fluctuating fluid velocity PDF is assumed to be a Gaussian probability distribution (Discrete Random Walk Model - DRWM).⁵ Thus, the particle velocity (v) at the particular location (*x*) is calculated by the set of equations presented below. In these equations, upper indices "new" and "old" describe a new velocity at a new position and an old velocity at an old position, respectively; Δt is the time step which the instantaneous fluid velocity is constant (e.g. eddy lifetime) and τ_p is the already mentioned relaxation time.

$$\mathbf{v}_{i}^{\text{new}} = u_{i} + \left(\mathbf{v}_{i}^{\text{old}} - u_{i}\right)e^{-\Delta t/\tau_{p}}$$
$$x_{i}^{\text{new}} = x_{i}^{\text{old}} + \frac{\Delta t}{2}\left(\mathbf{v}_{i}^{\text{new}} + \mathbf{v}_{i}^{\text{old}}\right)$$
(2.32)

Due to the high requirements of the computers storage and CPU time, only representative samples of the particle trajectories can be determined. On the other hand, the number of tracked particle trajectories should be sufficient in order to achieve a statistically independent solution - when an increase of number of particles does not have influence on the appearance of the particles at certain location. Thus realistic particle trajectories have been determined. For more details see Section 3.4.3.

⁵In most Lagrangian particles models which are based on particle momentum equation, the Monte Carlo method is involved in tracking the representatives samples of particle trajectories.

Crossing trajectory effect

As was explained above, particles moving in turbulent flows are "trapped" inside the eddy and they "jump" after the eddy decays. There is also the possibility that particles migrate to another eddy before the decay. This is known as the Crossing Trajectory Effect (CTE). In order to account for the CTE, the particle-eddy interaction time is required. Thus the minimum crossing time t_c is estimated according to Equation 2.33.

$$t_c = -\tau_p \ln\left(1 - \frac{l_e}{\tau_p |u - v|}\right) \tag{2.33}$$

The minimum crossing time is the time the particle would take to cross the eddy with characteristic length l_e . If the minimum crossing time t_c is smaller than the eddy lifetime τ_L , the particle would "jump" to another eddy. Hence, the eddy-particle interaction time would be taken as the time step in Equation 2.32.

2.4 Summary

This chapter gives an introduction to the simulations of gas-solid flows. The methods used for classifying these flows are presented and relevant parameters describing the flow of raw meal particles in the calciner are defined. There is also a description of a way of resolving the most important parameters by FLUENT.

3

Modeling

This chapter describes the main steps of modeling the flow of raw meal in the calciner. Firstly, a mesh creation is discussed and relevant aspects concerning designing the grid are brought up. Then, FLUENT settings are introduce. A description of discrete phase model including brief section about raw meal properties and stochastic independency are presented.

3.1 Grid design

There are several softwares used for meshing objects and no recommendation for the right choice can be given. This is because the grid made by using most of them can be very alike. The most important, when creating the grid is to provide an adequate resolution of the important flow features, as well as geometrical features. Thus, the grid must be fine enough, preferable made according to guidelines in, for example, FLUENT (2006), GAMBIT (2007), etc.

3.1.1 Mesh creation in Gambit

A model of the calciner was built and meshed in Gambit version 2.3.16. The grid is bodyfitted, structured and it based on hexahedral elements. In order to have the possibility of meshing the model, using hexahedral elements without any difficulties, the domain was subdivided on five subdomains - as it is shown on Figure 3.1. Each subdomain was meshed separately, but in the same manner. Neighbouring subdomains share a common wall and due to that fact these wall is treated as an interior. The duct delivering the tertiary air was meshed separately. At the intersection of the tertiary air duct and the calciner, an interface zone was created.

Firstly, two outer faces of each subdomain normal to the direction of the main flow (basically inlet and outlet) were meshed by using tool in Gambit called the *Map Scheme*. This scheme creates a regular, structured grid with quadrilateral or triangular mesh elements (the model of the calciner is only built of quadrilateral elements). Obviously, to control the mesh density, edges of meshed faces were previously marked with certain number of nodes depending on numbers of desired cells in each subdomain. There are 14 nodes on each edge of the meshed outer faces normal to the direction of the flow. It gives 980 cells for one face. Due to the large velocity gradient in the vicinity of the wall, the grid is more dense in that region in order to correctly resolve boundary layers. Important parameter when creating fine grid



Figure 3.1: Domain subdivided on five subdomains.

in the surroundings of the wall is the distance from the wall to a centroid of a cell adjacent to the wall, namely y^+ . Value of y^+ depends on the model used for modeling the near-wall region. In the simulations concerning present report, so called *Standard Wall Functions* were used. More detailed descritpion concerning boundary layers can be found in Section 3.1.2.

Having meshed faces at the inlet and the outlet (see Figure 3.1), subdomains were meshed by tool in Gambit called the *Cooper* scheme. It is based on sweeping the mesh nodes patterns of specified source faces (here faces at the inlet and the outlet) through the volume with the number of intervals describing the height of particular face. It has to be noticed that those intervals are not with the same length through the calciner. They differ due to the different diameter of calciner in various heights and the region of interested (upper part of the calciner and "swan neck") - Figure 3.4).

The duct used for delivering the tertiary air was meshed separately, using similar steps to those described above. The interface zone were created in a place of intersection of the wall of calciner and the wall of tertiary air duct. It means that all of quantities resulting from momentum, energy and mass equations are transferred from interface-adjacent cell of the duct to interface-adjacent cell of the calciner.

3.1.2 Boundary layers

A boundary layer is the layer of fluid in the immediate vicinity of a bounding surface. According to Versteeg and Malalasekera (2007), the boundary layer is composed of two regions: the inner region (10-20% of the total thickness of the wall layer) and the outer region, where the flow is not affected by viscous forces. Within the inner region are three zones listed below in order of increasing the distance from the wall *y* expressed by y^+ value. There is also an estimation of the dimensionless velocity in the main direction of the flow (U^+).

• **the viscous sublayer** - the flow is almost laminar, and the viscous stresses play a dominant role.

$$U^+ = y^+ \qquad y^+ \leqslant 5 \tag{3.1}$$

• the buffer layer - the viscous and turbulent stresses are of similar magnitude.

$$U^{+} = -3.05 + 5\ln y^{+} \qquad 5 < y^{+} < 30 \tag{3.2}$$

• the log-law layer (turbulent layer) - turbulent stresses dominate.

$$U^{+} = 2.5 \ln y^{+} + 5.45 \qquad y^{+} \ge 30 \tag{3.3}$$

The y^+ value and dimensionless velocity U^+ are defined as

$$y^{+} = \frac{u_{\tau}}{v}y \tag{3.4}$$

$$U^+ = \frac{u}{u_\tau} \tag{3.5}$$

where *u* is the velocity of the main flow, and u_{τ} is the so-called friction velocity which is

$$u_{\tau} = \sqrt{\frac{\tau_{xy}}{\rho}} \tag{3.6}$$

The wall shear stress τ_{xy} is described by Equation C.3. It must be noted that there is no velocity in the *y* direction ($\nu = 0$), thus the wall shear stress is reduced to $\tau_{xy} = \mu(du/dy)$. As it was mentioned, "wall functions" were applied to solve the boundary layers. These function are used to bridge the viscosity-affected region between the wall and the turbulent layer. Thus, according to FLUENT (2006), the y^+ value has to be in the range from 30 to 300, but preferable closer to the lower bound ($y^+ \approx 30$). Considering the flow in the calciner with the mean velocity equal approx. 2.5 (m/s) and the distance from the wall to the end of the cell adjacent to the wall which is equal 0.013 (m), the y^+ values for each cell of the meshed calciner, resolved for the flow with $r_t = 20\%$ are shown on Figure 3.2. This figure shows that y^+ is acceptable, thus the boundary layers are resolved correctly.



Figure 3.2: The y^+ values along the calciner.

3.1.3 Mesh quality

The grid was designed according to the guidelines presented in Casey and Wintergerste (2000) as well as according to the hints stated in FLUENT (2006) and GAMBIT (2007). The most common, describing the quality of the cells (a skewness, an aspect ratio of the sides and a change of the size of neighboring elements) are shown in 3.1.

| Quality type | Suggested Limit | No of cells above the limit (%) |
|----------------|-----------------|---------------------------------|
| EquiAngle Skew | EAS < 0.55 | 595 (0.18%) |
| Aspect ratio | AR < 5 | 1278 (0.38%) |
| Size change | <i>SC</i> < 2 | 252 (0.07%) |

Table 3.1: Example of the quality types of the cells.

This table also presents the number of cells of the meshed calciner, which do not meet the suggested criteria. These cells are mostly placed in the region of the intersection between the tertiary air duct and the calciner and in the vicinity of the bend of the "swan neck" -Figure 3.3. However, there are not that many elements above the limit of the quality type. The results are acceptable as the flow seems to be reasonable.



Figure 3.3: Regions of the bad-quality cells of the meshed calciner. Figure on the left: the bend of the "swan neck", figure on the right: the tertiary air duct.

Total number of cells of the grid used for simulation is 356,864 with the total number of nodes equal 348,903. Minimum and maximum volume of the cell is about $1 \cdot 10^{-7}$ (m³) and $2 \cdot 10^{-5}$ (m³), respectively. Meshed calciner is shown on Figure 3.4.



Figure 3.4: Final mesh of the calciner. Red lines present velocity profiles for grid-independence study.

3.1.4 Grid-independence study

It is relevant to make the grid-inpependence study in order to prove that solution is independent of the mesh used in simulations. To this end, two additional meshes were created with different number of cells. *Coarse mesh* is around 3 times less dense than the regular mesh described in Section 3.1 and it contains 120,140 cells. *Fine mesh* with 726,656 cells is about twice more dense than the regular mesh. Those meshes were made in exactly same manner as regular mesh with suitably different intervals between nodes. In both cases, the distance from the wall to the wall-adjacent cell (y^+) lies in the range suggested in FLUENT (2006).

The grid-independence study was carried out by simulating flow of the air in the calciner. In order to simplify simulations, only momentum equations were solved with the $k-\epsilon$ model simulating the turbulence (Section 2.1.1). The air was divided equally (r_t is 50%). Mass flow

rate for both, the tertiary air and the kiln gases was set as 0.5 (kg/s). The main difference between grids examined is the time of simulations which depends on a number of iterations required to reach desired convergence as well as number of cells. Simulations were stopped when residuals of the continuity equation become flat. The plot of the residuals in function of the iterations for the coarse mesh is shown on Figure 3.5. Characteristic jump represents change of the discretization scheme (from the *First Order Upwind* to the *Second Order Upwind*). The progress of the residuals for the regular and the fine mesh is very alike, except the number of iterations which is higher. Furthermore, the time of one iteration for the fine grid is much longer resulting in an increase in computation time used by 100% for a mesh twice more dense.



Figure 3.5: Residuals of the simulations carried out for the coarse grid.

Grid-independency study considering this report is based on the comparison of the velocity profiles at different heights of the calciner. It is commonly known that the larger the number of cells, the better the solution accuracy (Hefny and Ooka, 2009). Therefore, the results for the coarse grid and regular grid are compared to the dense grid's results which are the most realiable. Several profiles were juxtaposed - two of them are shown on Figure 3.6 and the rest can be found in Appendix D. Figure 3.6 presents plots of the velocity profiles in the upper part of the calciner right above the converging/diverging section at z = 2.895 (m) along the both x and y axises. Clearer view of the places of the velocity profile comparison is shown on Figure 3.4. Velocities for regular grid are very close to these for fine grid and in some places are exactly the same, in contrast to the velocities for coarse grid (the difference is significant in some regions). Therefore, the regular grid was used for further simulation.



Figure 3.6: Comparison of the velocity profiles for grids with different number of cells. Left plot: velocity profile in the calciner at z = 2.895 (m) along the *x* axis, right plot: velocity profile in the calciner at z = 2.895 (m) along the *y* axis

3.2 Raw meal properties

Raw meal is a mixture of solid particles which are firstly calcined in the calciner (release of CO_2) and later undergo several chemicall processes in the rotary kiln. Clinker is the product coming out from the rotary kiln and it is half-product of the cement. Raw meal is composed from limestone (CaCO₃), shal or clay (SiO₂, Al₂O₃, & Fe₂O₃) and some additives (SiO₂, Al₂O₃ or Fe₂O₃) - typicall proportion of these materials is shown in Table 3.2.

| Material | Amount (%) |
|--------------|------------|
| Limestone | 85% |
| Shal or clay | 13% |
| Additives | 1% each |

Table 3.2: Typical proportion of the materials composing a raw mixture (Alsop, 2005).

It is important that composition of raw mix is appropriate in order to obtain high quality cement, to avoid build-ups in the rotary kiln and the calciner, etc. An approximate analysis for raw meal on an ignited basis (without CO_2)¹ is presented in Table 3.3.

Various composition of the mineral materials (limestone, shale, clay) depending on the place where they are mined as well as various amount of additives added cause that properties of the raw meal used in several diverse cement plants can be significantly different. Properties used for the purpose of this project are briefly presented below inclusive an estimations and/or assumptions when final values were chosen.

¹CaCO₃ undergoes calcination causing release of CO₂ in quantity which is \approx 35% of total weight

| Chemical component | Amount (%) |
|--------------------------------|------------|
| CaO | 65-68% |
| SiO ₂ | 20-23% |
| Al_2O_3 | 4-6% |
| Fe ₂ O ₃ | 2-4% |
| MgO | 1-5% |
| Mn_2O_3 | 0.1-3% |
| Ti ₂ | 0.1-1% |
| SO ₃ | 0.1-2% |
| K ₂ O | 0.1-1% |
| Na ₂ O | 0.1 - 0.5% |

Table 3.3: Chemical composition of the raw mixture on ignited basis (Alsop, 2005).

3.2.1 Density

It was assumed that the specific density and the bulk density of raw meal used in experiments is approximately $\rho_s = 2900$ and $\rho_{bs} = 1400$ (kg/ m^3), respectively. This approximation is based on rough calculation of the mean density as a function of mass fraction and density of each component from Table 3.3 and estimation of void fraction for unspherical particles based on measurements described by Arngrímsson et al. (2008).

3.2.2 Sphericity

Sphericity is a measure of how spherical (round) an object is. Sphericity of the particle ϕ is defined as a ratio of the surface area of a sphere with the same volume as given particle to the surface area of this particle - Equation 3.7. The surface area of the sphere A_s is in a function of the volume of the particle (Ciborowski, 1965).

$$\phi = \frac{A_s}{A_p} = \frac{\pi^{1/3} \cdot (6V_p)^{2/3}}{A_p} \tag{3.7}$$

where V_p is volume of the particle and A_p is surface area of the particle. However, it is a difficult task to find a proper sphericity for particles as they have very unregular shapes - see Figure 3.7. This figure presents the snapshot from the microscope of the raw mixture with the characteristic dimension higher than 45 μ m. It can be seen that the shape of each particle is different what means that estimating just one value of the sphericity for all particles is a big approximation.

On the other hand, analyzing Figure 3.7 one can be assumed that the highest aspect ratio of the sides of the area seen is about $2.^2$ The sphericity of the rectangular cuboid with the aspect ratio of the one of its lateral area equal 2, and third dimension comparable with the shorter side of the lateral area, is approximately 0.77. This value can be taken as the lowest

²Picture from the microscope is two-dimensional, but according to the physics it is reasonable assumption that third dimension is not the longest.



Figure 3.7: Microscope view of the raw meal particles with diameter greater than 45 μm . Measurements were carried out in FLSmidth R&D Centre Dania.

sphericity of the raw mixture particles. Next, the drag coefficient for different values of the sphericity is calculated (Equation 3.8) and depicted on Figure 3.8. Only the drag force is considered in this analysis as this is the major force which acts on the particles moving in the fluid (see page 13). The drag coefficient for non-spherical particles with various sphericity is estimated by following equation (Haider and Levenspiel, 1989):

$$C_{D} = \frac{24}{Re_{p}} \left(1 + e^{2.3288 - 6.4581\phi + 2.4486\phi^{2}} Re_{p}^{0.0964 + 0.5565\phi} \right) + \frac{Re_{p}e^{4.905 - 13.8944\phi + 18.422\phi^{2} - 10.2599\phi^{3}}}{Re_{p} + 1.4681 + 12.2584\phi - 20.7322\phi^{2} + 15.8855\phi^{3}}$$
(3.8)

The drag coefficient for a sphere becomes (Rosendahl, 1998):

$$C_D = \frac{24}{Re_p} \left(1 + 0.1806 Re_p^{0.6459} \right) + \frac{0.4251}{1 + \frac{6888.95}{Re_p}}$$
(3.9)

The drag coefficient for sphere and for non-spherical particle with the sphericity $\phi = 0.77$ is comparable - the difference is not higher than 10% for $Re_p < 2$. Furthermore, it lies within the range of the error of drag coefficient for non-spherical particles which is accounted for approximately 20% (Haider and Levenspiel, 1989), as it is shown on Figure 3.8. Therefore, in this project, particles of raw meal are treated as spheres ($\phi = 1$).

3.2.3 Particle size distribution

Size of the particles is a very important parameter when simulating multiphase systems. It can be seen on Figure 3.7, that raw meal is composed of many different sizes of grains. Therefore, Particle Size Distribution (PSD) was measured by particle size analyzer called *MasterSizer*



Figure 3.8: Drag coefficient for non-sherical particles with various sphericities. Both plots are for the same conditions; on the right plot the drag coefficient for sphere is compared with the drag coefficient for non-spherical particle with $\phi = 0.77$.

2000. This device is based on laser diffraction technology. Measurements were carried out in FLSmidth R&D Centre Dania. The scan of the measurement's raport as well as short description of the operation of *MasterSizer 2000* and assumptions made in this technique can be found in Appendix E.

It is possible to simulate gas-particle flow in FLUENT with using various sizes of particles as long as PSD is described in form of Rosin-Rammler type (FLUENT, 2006):

$$Y_d = e^{-\left(\frac{dp}{d}\right)^n} \tag{3.10}$$

where Y_d is the mass fraction of the particles of diameter greater than d_p , \overline{d} is the mean diameter of the probe of solids measured and n is so called spread parameter, which is constant. Therefore, the PSD obtained from the *MasterSizer 2000* were transferred to the Rosin-Rammler type of the particle distribution. Then the mean particle diameter was calculated. There are several different types of the mean diameters which can be found in the literature, but for the sake of using Rosin-Rammler type of PSD, so called d_{50} were chosen. Definition of the diameter d_{50} says that 50% of the entire mass of the probe is composed of particles with the diameter greater than the mean particle diameter. Finally, spread parameter n was found by minimising the difference between the mass fraction Y_d measured and calculated. Results are presented on Figure 3.9. It was assumed that diameters of particles injected to the calciner are in the range from 1 to 100 μm - as it is marked on Figure 3.9. Twenty different diameters from this range were used in the simulations. Parameters describing the Rosin-Rammler distribution of the raw meal particles are listed in Table 3.4.



Figure 3.9: Particle size distribution (Rosin-Rammler type). Plot presents comparison of measured and calculated particle size distribution.

| Min. diameter | 1 <i>µ</i> m |
|------------------|---------------|
| Max. diamter | $100 \ \mu m$ |
| Mean diameter | 13 µm |
| Spread parameter | 0.85 |
| No. of diameters | 20 |

Table 3.4: Parameters describing the Rosin-Rammler distribution.

3.3 Feeding of raw meal

The raw meal is transported into the calciner through pipes connecting a feeder and a spreader box. There is an airlock rotary valve between minimizing the air leakage. This is shown in Figure 3.10.

3.3.1 Raw meal injection

Although, the airlock rotary valve was used, there is a noticeable amount of the air transported with the raw meal (rough calculation are shown below). It means, that particles flow with a similar velocity as the air. The experiments done by Borawski (2008) shows that raw meal enters the calciner mostly from the bottom part of the spreader box. Therefore, it was assumed that raw meal particles are injected in two ways: three-quarters of the total amount is fed from the bottom of the spreader box and one fourth from the upper part. The height of the upper part is twice as long than the height of lower part ($h_u = 2h_d$). This is shown schematically on Figure 3.11.

The particles entering the calciner from the down part of the spreader box move in the direction resulting from the angle between the adjustable plate in the SB and the horizontal



Figure 3.10: The feeding system of raw meal. The drawing on the left was made by A. Einarsson (FLSmidth's engineer). The picture on the right shows the spreader box used in the experiments carried out by Borawski (2008).



Figure 3.11: Raw meal injection into the calciner.

plane, which is $\alpha = 30^{\circ}$. However, the rest of the particles are injected from upper part in the direction normal to the wall of the calciner. In FLUENT simulations concerned this project, particles are injected from a surface, which is basically the area of SB wall adjacent to the

calciner. In order to simplify, the arch at the bottom of this surface (resulting from the shape of the plate in the SB) is simply a straight line. Thus, this surface was defined in FLUENT in the way presented on Figure 3.12.



Figure 3.12: The surface of release the raw meal particles defined in FLUENT.

3.3.2 Leakage air

Next step is to determine a velocity of the raw meal fed into the calciner, which is based on a velocity of the leak air. It was assumed that the air transported together with raw meal is basically the air contained in the rotary chamber of the rotary valve and the air coming from a leakage from badly sealed or non sealed places. This is shown schematically on Figure 3.13.



Figure 3.13: The air leakage through the rotary valve.

The volume of the rotary chamber comprised of 6 compartments is equal

 $V_{rc} = \pi \cdot 0.22 \cdot 0.11 \implies V_{rc} = 0.076 \,\mathrm{m}^3$

The rotary chamber makes one complete revolution over approximately 3 (s). This means, that two compartments are filled with raw meal (and air) during one second. The volume of raw meal transported in one compartment is negligible (approx. $3 \cdot 10^{-5} m^3$). Thus, the airflow transported through the rotary chamber is equal 0.0253 (m³/s). Assuming that half of this amount leaks from other places (for example the way shown on Figure 3.13), the total flow of the leakage air is $V_{la} = 0.038$ (m³/s) which is 4% of the total airflow.

3.3.3 Parameters of raw meal injection

Raw meal particles are rather small (Section 3.2.3) and their velocity in the pipe between the rotary valve and the spreader box is almost the same to the velocity of the leakage air, which can be estimated as

$$u_{la} = \frac{4 \cdot 0.038}{\pi \cdot 0.08^2} \quad \Rightarrow \quad u_{la} = 7.5 \,\mathrm{m/s}$$

The final velocity of raw meal is lower, because particles lose their momentum due to the friction forces, the bend of the pipe, an impact with the adjustable plate, etc. Thus, it was assumed that at the bottom of the SB raw meal enters the calciner with the velocity equal 5 (m/s). In order to stabilize the simulations, another assumption was made, namely the leakage air was injected to the calciner. The flow of leak air was divided exactly in the same way as the surface of the raw meal injection - the air velocity flows through the bottom of the SB is 5 (m/s). The inlet conditions of the remaining particles (and the air) for upper part of the SB result from the difference betwen the total flow of the leakage air and the airflow through down part of the SB. Table 3.5 shows the final inlet parameters defined in FLUENT. It has to be mentioned, that velocity components result from the angle α and relate to the coordinates system shown on Figure 3.11.

| Parameter (unit) | Part of the spreader box | |
|----------------------------------|--------------------------|---------|
| | Down | Upper |
| Area (m ²) | $4.77 \cdot 10^{-3}$ | 0.0106 |
| Hydraulic diameter (m) | 0.063 | 0.103 |
| Mass flow (kg/s) | 0.063 | 0.021 |
| Velocity magnitude (m/s) | 5 | 1.3 |
| Velocity components (m/s) | u = 0 | u = 0 |
| | v = 4.3 | v = 1.3 |
| | w = -2.5 | w = 0 |

Table 3.5: The inlet conditions of raw meal injection defined in FLUENT

3.4 Settings in FLUENT

The simulations were carried out in FLUENT version 6.3.26. The simulations were run on steady-state mode. *First Order Upwind* discretization scheme was used to solve the governing equations with the pressure-based solver. In order to stabilize the simulations, the underrelaxation factor was changed for the energy equation from 1 to 0.99 and for the discrete phase model from 0.5 to 0.2. The $k - \varepsilon$ turbulence model was applied with the default settings (see Section 2.1.1). Approximately 5,000 iterations were made for each simulation. A typical plot of residuals obtained for each simulation made is shown on Figure 3.14.



Figure 3.14: Progress of residuals for simulations carried out for the flow with raw meal particles.

3.4.1 Boundary conditions

The boundary conditions set in FLUENT are the same like those for the experiments carried out by Borawski (2008), presented in Table 1.1. It was assumed that the tertiary air and the kiln gases enter the calciner with a uniform velocity profile. The turbulent intensity at both inlets is 8%. It was assumed that the wall of the calciner (made from Plexiglass) has a constant temperature of $T_w = 60^{\circ}$ C. The properties of the air (density, viscosity and thermal conductivity) are calculated from the linear interpolation of the data taken from Çengel (2007). Although, there is an underpressure in the calciner, it was assumed that it is negligible - thus the atmospheric pressure is used.

3.4.2 Discrete Phase Model (DPM)

Discrete Phase Model is a model used for simulating two-phase flows according to the description in Section 2.2. The main limitation of this model is that a dispersed phase is sufficiently dilute - volume fraction lower than 10-12% (FLUENT, 2006). This limitation results from the neglecting the particle-particle interactions and the volume occupied by particulate phase. According to the calculations presented in Section 2.3, the DPM model can be used for modeling the gas-solid flow concerning present report. DPM settings used in the simulations concerning the flow of the raw meal particles and the calciner are described briefly below.

Interaction between continuous and dispersed phase

The interaction between the air and the raw meal is enabled with one DPM iteration per 50 iterations of the continuous phase. The particle tracking is done in steady-state mode. The *Step Length Factor* is set to 3. It means that a time step used to integrate the equation of motion for particles is three times lower then a time required for these particles to traverse a controll volume (a computational cell of the meshed calciner).

Drag Parameters

According to FLUENT (2006), a high-Mach-number drag law is recommended if Ma > 0.4, where Ma is the Mach number expressed as

$$Ma = \frac{\mathbf{v}}{u_s} \tag{3.11}$$

Symbol u_s is the speed of sound in a medium. The speed of the sound in the dry air at $T = 80^{\circ}$ C can be estimated by $u_s \approx 20\sqrt{T}$, which is approx. 375 (m/s). Thus, the Mach number for the particles moving in the calciner is very small ($Ma \ll 0.1$).

The Cunningham correction to drag law needs to be applied for such a small particles, that the fluid may no longer be considered continuous. The Cunningham correction factor C_c for air at ambient conditions is expressed by

$$C_c = 1 + \frac{2\lambda}{d_p} \left(1.257 + 0.4e^{\frac{-1.1d_p}{2\lambda}} \right)$$
(3.12)

where λ is the molecular mean free path of the air molecules, which is approx. 65 (nm). The Cunningham factor for raw meal particles with diameter $d_{50} = 13$ (μ m) is $C_c = 1.013$. Thus, the Cunningham correction is not enabled for the simulations.

Accordingly to the calculations made in Section 3.2.2, raw meal particles can be treated as spheres. Thus the equation of motion is based on drag forces derived for spheres.

Physical models

There are an optional forces and effects concerning motion of dispersed phase available in FLUENT. They are presented in Table 3.6 where a short description is given whether force is enabled or disabled in the simulations.

| Force | Usage | Description |
|----------------------|----------|--|
| Brownian force | disabled | Only for very small sub-micron particles (Kær, 2001) |
| Thermophoretic force | disabled | The temperature gradient is too low |
| Saffman lift force | enabled | May play important role close to the wall |
| Erosion/Accretion | disabled | PSD of raw meal is the same before and |
| | | after experiments (Figures E.1 and E.2) |
| Two-way coupling | enabled | Mutual effect between phases (see Section 2.3.1) |

Table 3.6: Optional forces and effects affecting particles

Injections

Detailed description of feeding raw meal can be found in Section 3.3. Raw meal particles are injected into the calciner from two surfaces (down and upper part of the SB). The initial conditions for these injections are shown in Table 3.5. Twenty various diameters from the range described by Rosin-Rammler distribution (Table 3.4) are released from each cell of the injection surface (Figure 3.12). Total number of particles tracked depends on number of tries of the discrete walk random model. It was assumed that 50 tries gives a statistically independent solution (see Section 3.4.3). There are 24 cells in the surfaces, where raw meal particles with 20 various diameters are released from. This gives 24,000 particles, which are tracked during the simulations.

3.4.3 Statistical-independence study

A total number of particles moving in the calciner can be estimated in the following way:

- a volume of one particle with the mean diameter $d_{50} = 13 \ \mu \text{m}$

$$V_p = \frac{\pi d_p^3}{6} \Rightarrow V_p = 1.15 \cdot 10^{-15} \,(\text{m})$$

- a volume of raw meal injected into the calciner per second

$$V_{rm} = \frac{\dot{m}}{\rho_s} \Rightarrow V_{rm} = 2.9 \cdot 10^{-5} \,(\mathrm{m}^3/\mathrm{s})$$

- a number of particles injected into the calciner per second

$$n_t \cdot V_p = V_{rm} \implies n_t = 2.5 \cdot 10^{10} \text{ (particles/s)}$$

An average residence time of the particles in the calciner is approx. 1.5 (s) (see Section 4.3). Hence, $3.75 \cdot 10^{10}$ particles are moving in the calciner in each moment. In general, it is not feasible to track each and every particle due to high computational demands. Instead a number of representative particle trajectories are tracked which represent a packet of particles with the same sizes and initial parameters. Obviously, the higher number of particles tracked, the more realistic solution. There are at least two reasons of that fact. Firstly, the particles represent given mass flow. This means that smaller number of particles represent higher mass

3 Modeling

of particles, what can cause a problem with a convergence. Secondly, due to the stochastic nature of an eddy lifetime concept - fluctuating velocity components are determined based on a Gaussian probability distribution (see paragraph 2.3.4) - the large number of particle trajectories are needed in order to obtain statistically independent representation. The number of particles also depends on a mesh density - the more cells in a cros-section, the larger number of particles required. A comparison of particle concentration at different heights of the calciner for different number of released particles is shown and analyzed below.

Simulations were carried out for four numbers of particles: 2,400, 9,600, 24,000, and 48,000. The comparison of a particle concentration for different numbers of tracked particles is shown on Figure 3.15. As was mentioned, a "better" result is obtained for larger amount of particles, thus simulations with 48,000 particles are the most proper - the curve is rather smooth comparing to the others. The particle concentration of the simulation carried out for 2,400 particles is characterized by strong fluctuation and values are rather not comparable with these for 48,000. "Better" results are obtained from the simulations made for 9,600 and 24,000, but the latter shows the highest similarity in most places analyzed (see figures in Appendix F). A computational time required for simulating a flow of raw meal particles in the calciner with 24,000 tracks is much lower than the time needed for 48,000. Thereby, it was assumed that the simulations carried out for 24,000 particles gives the statistically independent solution. On the other hand, this amount is approx. 0.00006% of the total number of particles in the calciner. Thus, in case of a very detailed analysis, higher number of tracks should be considered.



Figure 3.15: Comparison of a particle concentration for different number of particles at z = 3.495 (m). The left plot: across the *x* axis, the right plot: across the *y* axis.

3.4.4 Heat transfer between phases

The heat transfer associated with the raw meal particles in the calciner can be determined from the following formula:

$$\dot{Q} = n_t h S_p \left(T_a - T_{rm} \right) \tag{3.13}$$

where S_p is the surface of one particle and h is the heat transfer coefficient. The raw meal particles are very small, therefore it can be assumed that the temperature is uniform throughout the particle - the Biot number is low (see clarification below). The heat transfer coefficient is determined from the definition of the Nusselt number which is

$$Nu = \frac{hd_p}{k_a} \tag{3.14}$$

where k_a is the thermal conductivity of the air. The Nusselt number for flow over a sphere for very low particle Reynolds numbers ($Re_p < 1$) approaches the theoretical value for the stationary fluid, which is Nu = 2 (Kunii and Levenspiel, 1991). Thus, the heat transfer coefficient for the particle with mean diameter $d_{50} = 13 \,\mu\text{m}$ is approx. $h = 4,540 \,(\text{W/m}^2\text{K})$ - the air properties are taken for its highest temperature ($T_a = 80^{\circ}\text{C}$). As the heat transfer coefficient is known, the Biot number expressed by Equation 3.15 can be calculated.

$$Bi = \frac{hd_p}{k_s} \tag{3.15}$$

where k_s is the thermal conductivity of the raw meal. The raw mixture is mainly composed of calcium carbonate, and the thermal conductivity can be taken for this material ($k_s = 2.25 \text{ (W/m-K)}$ - the value taken from the FLUENT database). Thus, the Biot number for mean particles is approx. 0.026. It means that assumption of uniform temperature throughout the particles is correct (as long as Bi < 1).

The heat transferred in one second from the air to the particles calculated according to Equation 3.13 is $\dot{Q} \approx 3.194 \cdot 10^6$ (W). Due to the small sizes of particles, the time required for heating the particles is much lower than one second. This time relates to the responsiveness of a particle to changes in temperature in the carrier fluid (an analogy to the momentum response time - page 2.3.2) and this is the so called thermal response time. In other words, the thermal response time τ_T is the time required for a particle to achieve 63% of the fluid temperature and is expressed as follows (Crowe et al., 1998):

$$\tau_T = \frac{\rho_s C p_s d_p^2}{12k_a} \tag{3.16}$$

where Cp_s is the specific heat of the particles. The raw mixture is mainly composed of calcium carbonate, and the specific heat can be taken for this material ($Cp_s = 856$ (J/kg·K) - the value read from the FLUENT database). Table 3.7 shows the thermal response time for smallest, mean and largest raw meal particles moving in the calciner.

Hence, assuming that the raw mixture consists of particles with mean diameter d_{50} , the heat transfer between the air and the number of particles injected into the calciner in one second is

$$Q = \dot{Q} \cdot \tau_T \quad \Rightarrow \quad Q = 3800 \, \text{(J)}$$

3 Modeling

| Size (µm) | $	au_T$ (s) |
|-----------|----------------------|
| 1 | $7\cdot 10^{-6}$ |
| 13 | $1.19 \cdot 10^{-3}$ |
| 100 | 0.07 |

Table 3.7: The thermal response time for raw meal particles moving in the calciner.

3.5 Summary

4

Results

This chapter contains a description of a flow of raw meal particles in the calciner based on simulations carried out in FLUENT. Two cases (1 and 2) with different ratio of a tertiary air and kiln gases are analyzed and compared regarding raw meal distribution. Case 1 is the flow with a low ratio of the tertiary air to the total flow equals $r_t = 20\%$. Case 2 is the flow with the high ratio, which is $r_t = 55\%$. Finally, a comparison of the simulations and an experimental data carried out by Borawski (2008) is shown.

4.1 Flow features

The main features for both types of the flow (case 1 and case 2) are shown on Figure 4.1, where the velocity magnitude is displayed on tranverse contours. The figure shows that the flow is quite different in both cases. When analyzing the case 1 (lower tertiary airflow), it can be observed that the kiln gases determine the flow - there is a core with high velocity in the centre of the bottom part of the calciner - see Figure 4.2. The tertiary air entering the calciner causes the core to be "moved" from the center to the opposite wall of the tertiary air inlet along the lower part of the calciner (region marked with 1 on Figure 4.1). Thus, the rest of the air (the tertiary air) flows in the vicinity of the walls opposite to the mentioned core with a very low velocity. There is also observed a region with a backflow (marked with 2). Clearer view of this flow pattern is shown on Figure 4.3, where the velocity magnitude is displayed in form of vectors.

The velocity profile for the case 2 (the flow with dominating tertiary air) is entirely different. Firstly, this flow is highly turbulent - the fluid velocity components vary significantly. This pattern can be observed on figures presenting velocity vectors (see Figure 4.2 and Figure 4.3). The tertiary air flows with high velocities in the vicinity of the walls - the crescent-shaped region with a gradually decreasing velocity field from the wall to the center. There is a vortex surrounded by this "crescent" (see Figure 4.1). Both, the vortex and the "crescent" rise spirally showing a revolving flow. However, the velocities "inside" the vortex are rather low and in the middle a reversed flow is observed (marked with 3). The reversed flow vanishes at approx. z = 1.5 (m), but there still is a region with low velocities. Clearer view is shown on Figure 4.3.



Figure 4.1: The velocity magnitude displayed on transverse contours for two types of the flow. The left figure: case 1, the right figure: case 2. The colorbar corresponds to the velocity magnitude.

Figure 4.2 shows the velocity profile (in a form of vectors) in the lower part of the calciner. In case 1, it can be observed that the velocity in the center is high and rather uniform, mainly in upward direction. The closer to the wall, the lower is the velocity. An inverse phenomena is observed in case 2. The highest velocities are in the vicinity of the walls in a direction resulting from the position of the tertiary air inlet. As was mentioned above, the vortex is created and the backflow occur right above the height of the tertiary air inlet.



Figure 4.2: The velocity vectors at the bottom of the calciner for two types of the flow. The left figure: case 1, the right figure: case 2. The colorbar corresponds to the velocity magnitude.

Figure 4.3 shows velocity vectors of the lower cylindrical part of the calciner. The colorbar corresponds to the axial velocity. It means that regions marked with blue and dark blue colors represent a reversed flow. The flow pattern in both cases is quite similar to each other - the velocities are low on the side of the tertiary air inlet and high on the opposite side. In both cases, there is also a backflow. However, in case 2 ($r_t = 55\%$) the flow is more chaotic (turbulent) - the local velocity components are various. The region of low velocities (the vortex) develops in the middle of the lower cylindrical part of the calciner. At the end of this part, the vortex is almost entirely dissipated.



Figure 4.3: The velocity vectors of the lower cylindrical part of the calciner for two types of the flow. The colorbar corresponds to the axial velocity.

4 Results

4.1.1 Effects of raw meal injection

Raw meal injected into the calciner strongly affects the flow. It can be seen from Figure 4.1, that there is a low-velocity region in the surroundings of the spreader box (in both cases). Analyzing case 1, it can be observed a high-velocity region (with nearly uniform velocity - see Figure 4.4) opposite to the raw meal inlet and a region with low velocities and a slight backflow (marked with 4 on Figure 4.1 and shown on Figure 4.4). The low-velocity region is a result of the raw meal injection. This flow pattern (two regions with high and low velocities) is very similar along the whole upper part of the calciner. At the end of this part, the flow becomes more uniform, but there still is a separation on two regions mentioned above. Basically, the places with low-velocities show high raw meal concentration.

Similarly, in case 2 the velocity is very low in the vicinity of the spreader box. However, the swirling effect causes the low-velocity region (or raw meal particles) to be "moved" to the center of the calciner along the upper part. The flow pattern in the upper part is quite similar to one in the lower part of the calciner - there is also a crescent-shaped region. Clearer view of the velocities in the upper part is displayed on Figure 4.4.



Figure 4.4: The velocity vectors of the upper cylindrical part of the calciner for two types of the flow. The colorbar corresponds to the velocity magnitude.

According to Figure 4.4 for $r_t = 55\%$, the flow is strongly swirling right after a converging/diverging section. Similar pattern (but with much weaker swirling effect) is observed for case 1. This is the effect of using the converging/diverging section, which gives rise to a secondary swirling flow. However, the strong swirling effect is damped by the raw meal particles (see the difference between the planes at heights z = 2.7 and z = 3.1 (m) on Figure 4.4 for $r_t = 55\%$). The raw meal injection causes the low velocities in the region close to the spreader box. Although, the strong swirling effect is dumped, the low-velocity region is "moved" according to the direction of the swirl. However, in the middle of the upper cylindrical part, the velocity magnitude becomes nearly uniform (with slightly lower velocity in the center) and the swirling effect is rather weak (marked with 5 on Figure 4.1).

Although the flow pattern is different for both cases, the difference in the swan neck is insignificant, especially in the part after the bend. In case 2, the swirling effect is nearly entirely dumped, the velocities are rather uniform in the cross-sections. In both cases, there is a backflow right after the bend (marked with 6 on Figure 4.1). The flow pattern in the "swan neck" is very alike in both cases.

4.2 Raw meal distribution

Distribution of raw meal particles in the calciner resolved from the simulations is displayed in a set of contour plots on Figure 4.5. These plots shows the concentration of particles (kg/m^3) in various cross-sections (places of the experiments (see Figure 1.2)). Figure 4.5 shows that raw meal distribution is different for both cases. It seems that particles are distributed better for case 1. This is already seen right after the spreader box (z = 2.895 (m) on freffig:uffff), where an area occupied by the particles is larger. However, as was described in Section 4.1, there is a region with very low velocities (also a reversed flow) in the vicinity of the wall above the spreader box resulting in a high concentration of raw meal in that area. Similar phenomena is observed in upper parts. Furthermore, the region with high particle concentration close to the wall is getting wider along the upper part of the calciner. The rest of the particles is quite well distributed within the area from the region of high concentration to the center of the calciner.

In case 2, it is observed the particles are to follow the main stream determined by the swirl. The region of high concentration rise spirally according to the flow pattern. Although the flow is highly turbulent the raw meal is not spread within the calciner in the lower parts - basically, the entire cluster move with the flow without mixing. In upper parts, particles are more evenly distributed within the area from the region of high concentration (in the vicinity of the wall) to the center.



Figure 4.5: Raw meal distribution displayed on contour plots for both types of the flow. The colorbar corresponds to a concentration of raw meal (kg/m^3) .

4.3 Residence time

An important parameter when analyzing a distribution of raw meal particles in the calciner is a residence time. This is an average time the particles spend within the calciner. The residence time for both flows analyzed in this report is presented on Figure 4.6 in form of histograms. These histograms shows a number of particles represented by a certain residence time. The number of particles is expressed by percentage of the total amount of particles tracked in the simultions.



Figure 4.6: Residence time of the raw meal particles in the calciner for two types of the flow. The left histogram: case 1, the right histogram: case 2.

In both cases, the residence time is similar and for most of the particles is around 1.5 (s). However, in case 2 ($r_t = 55\%$), the number of the particles in a function of the residence time is nearly evenly distributed. Most of the particles are present in the calciner over the time from 1 to 2 (s). There is a small amount of raw meal with the residence time slightly longer than 2 (s). Some of the particles moving under the flow with low ratio ($r_t = 20\%$) need more time to leave the calciner - significant fraction of raw meal spend up to 4 (s). The reason is that region of high raw meal concentration is in the vicinity of the walls, where the air flows with rather low velocity (see Section 4.2 and Figure 4.5). There are few particles, which are temporary "trapped" in the calciner (probably in the low-velocity regions) - thus, the residence time is up to 13 (s).

4.4 Experiments

Experiments concerned an investigation of raw meal distribution in the calciner carried out by Borawski (2008) are based on temperature measurements. A model of the calciner placed in FLSmidth R&D Centre Dania is the cold model (only the so called "cold tests" can be carried out). The experiments are based on injecting preheated air (approx. 80°C) and cold raw meal (room temperature). The temperature measured in various places gives a view on how the raw meal particles are distributed in the calciner. Basically, low temperature means high concentration and vice versa (see Section 4.5). The temperature was measured through placing a rod with thermocouples in the calciner as it is shown on Figure 4.7.



Figure 4.7: The rod with thermocouples in the calciner.

The measurements were carried out in four different heights: 2.895, 3.195, 3.495, and 3.795 (m) - see Figure 1.2). There are seven thermocouples placed in the rod. The distance between them is equal, which is 0.1 (m). The temperature was measured in four "lines" for one height. These lines are symmetrical - angle between each one is 45°. The temperature was measured in 25 points for each cross-section (7 thermocouples times 4 lines, while the temperature in the middle was measured 4 times). A cross-section of the calciner with measuring points and relevant data is shown on Figure 4.8.


Figure 4.8: Coordinates of thermocouples placed in the calciner during the experiments.

4.5 Relation between temperature and concentration

As was already mentioned, raw meal distribution in the calciner can be determine by measuring and analyzing the temperature in various places. It is rather difficult to obtain an exact pattern of raw meal distribution from these tests. However, it is possible to determine regions with high and low particle concentration. The main idea is that lower temperature shows high raw meal concentration and vice versa. This relation is displayed on Figure 4.9 for both types of the flow, where temperatures and raw meal concentration, both resolved from the simulations are plotted. The data correspond to the "line D" for z = 3.495 (m) - see Figure 4.8.



Figure 4.9: Relation between temperature and raw meal concentration.

4.6 Simulations vs experiments

The results from the simulations can be compared to these from the experiments by plotting the temperature in the corresponding points shown on Figure 4.8. The temperature profiles in the calciner at z = 3.795 (m) for case 1 ($r_t = 20\%$) are juxtapositioned on Figure 4.10 and 4.11. In order to get the whole view of raw meal distribution by analyzing the temperature profiles, the reader is reffered to Appendix G. In case 1, it is observed that the distribution of particles for the simulations and the experiments is quite similar - progress of the temperature profiles is comparable in some plots.



Figure 4.10: Comparison of the temperature profiles for the experiments and the simulations. Data correspond to the lines A and B for case 1 at z = 3.795 (m).



Figure 4.11: Comparison of the temperature profiles for the experiments and the simulations. Data correspond to the lines C and D for case 1 at z = 3.795 (m).

In case 2 (z = 3.795 (m)), the temperature profiles are juxtapositioned on Figure 4.12 and 4.13. The rest plots are placed in Appendix G. These figures show the raw meal is distributed differently in the calciner than it is predicted in the simulations. When analyzing the experimental curves, it seems that raw meal is mainly concentrated in the centre of the calciner, whilst the simulations curves show high concentration in the vicinity of the walls.



Figure 4.12: Comparison of the temperature profiles for the experiments and the simulations. Data correspond to the lines A and B for case 2 at z = 3.795 (m).



Figure 4.13: Comparison of the temperature profiles for the experiments and the simulations. Data correspond to the lines C and D for case 2 at z = 3.795 (m).

However, it is rather difficult to determine regions with high and low raw meal concentration based on these plots. Moreover, there are some uncertainties regarding correctness of the experiments and the simulations *i.e.* the temperature measured in the experiments is higher than the temperature resolved from the simulations. This fact is observed especially in regions of high raw meal concentration, where the temperature should be much lower. A justification of this phenomena can be that the rod with the thermocouples was already heated up when temperature was measured. Thereby, the measurement was affected due to radiation. It is also possible, that settings in FLUENT do not correspond to the reality *e.g.* the amount of cold air sucked through the spreader box.

4.6.1 Comparison of raw meal distribution based on contour plots

Results from the experiments and the simulations can be shown on contour plots. This method is based on a calculation of a parameter (it will be called a *weight factor*), which describes the amount of raw meal presented in certain area. The weight factor f_n is derived according to

the following procedure. The heat balance between an air and a raw mixture can be expressed as

$$T_{mix}(\dot{m}Cp_s + \dot{m}_a Cp_a) = \dot{m}Cp_s T_{rm} + \dot{m}_a Cp_a T_a$$
(4.1)

where Cp_s is the heat capacity, \dot{m} is the mass flow and T_{rm} is the initial temperature of raw meal. Similarly, Cp_a , \dot{m}_a and T_a are paramters of the air. T_{mix} is the temperature of a mixture composed of the air and the raw meal. As was described in Section 4.4, there are 25 measuring points in a cross-section. It can be assumed that the cross-section is divided equally on N = 25 cells, each represented by the measuring point. Thus, the same amount of the air flows through each cell. Thereby, Equation 4.1 can be rewritten for the n-cell:

$$T_n\left(f_n\dot{m}Cp_s + \frac{1}{N}\dot{m}_aCp_a\right) = f_n\dot{m}Cp_sT_{rm} + \frac{1}{N}\dot{m}_aCp_aT_a \tag{4.2}$$

where T_n is the temperature measured during the experiments. In order to simplify the equation above, a new parameter *K* is introduced

$$K = \frac{\dot{m}_a C p_a}{\dot{m} C p_s} \tag{4.3}$$

Hence, combining and rearranging Equation 4.3 and Equation 4.2, the weight factor is expressed as follows

$$f_n = \frac{K}{N} \frac{T_a - T_n}{T_n - T_{rm}} \tag{4.4}$$

The weight factor f_n is a measure of a raw meal fraction occupying an area represented by measured temperatures. Basically, the higher value of the weight factor, the higher is the concentration of raw meal particles. The contour plots for both cases ($r_t = 20\%$ and $r_t = 55\%$) are shown on a set of figures below. The *x* and the *y* axes correspond to coordinates shown on Figure 4.8. Figure 4.15 and Figure 4.14 shows the weight factor calculated for both cases ($r_t = 20\%$ - the left side and $r_t = 55\%$ - the right side) at z = 2.895 and z = 3.195 (m), respectively. As was already observed in previous analysis of case 1, a distribution of raw meal particles obtained from the experiments and the simulations (see Figure 4.5) is similar. For z = 2.895 (m), the particles mostly occupy a region in the vicinity of the spreader box. The concentration is gradually decreasing to the center of the calciner and in directions to the walls perpendicular to the spreader box. At z = 3.195 (m), a high concentration still is observed in the surroundings of the walls and gradually decreases.

In case 2 for z = 2.895 (m), most particles occupy a region between the wall and the centre of the calciner. Similarly to the results from the simulations (Figure 4.5), this region is "moved" by the air with respect to the spreader box. However, the disperion pattern at z = 3.195 (m) is quite different. The particles are mainly placed in the centre of the calciner. There is one similar feature of both, experimental and simulation patterns, that raw meal is not spread within the calciner and particles flow as a bunch.



Figure 4.14: The contour plots of the weight factor for both types of the flow. The plot on the left: case 1, the plot on the right: case 2.



Figure 4.15: The contour plots of the weight factor for both types of the flow. The plot on the left: case 1, the plot on the right: case 2.

Figure 4.17 and Figure 4.16 shows the weight factor calculated for both cases at z = 3.495 and z = 3.795 (m), respectively. In case 1, it seems that particles are more spread within the calciner in the upper parts. However, there is the region of high concentration in the surroundings of the wall - it is also observed on Figure 4.5, where the results from the simulations are shown. It is worth to mention about spots with high raw meal concentration. Their position in the cross-section of the calciner is nearly the same for the experiments and the simulations.

The dispersion pattern for case 2 at z = 3.495 (m) is nearly the same like the one described above for z = 3.195 (m), namely the particles occupy an area in the centre of the calciner. But, in the upper parts (z = 3.795 (m)), the particles are "pushed" to the wall - similarly to the results from the simulations. However, the region of high concentration still is observed in the surroundings of the centre.



Figure 4.16: The contour plots of the weight factor for both types of the flow. The plot on the left: case 1, the plot on the right: case 2.



Figure 4.17: The contour plots of the weight factor for both types of the flow. The plot on the left: case 1, the plot on the right: case 2.

4 Results

4.7 Summary

4 Results

5

Conclusion and discusion

This project concerns a numerical investigation of gas-solid flow in a calciner. The main objective was to compare results from simulations to these from experiments carried out by Borawski (2008). The experimental work concerned an investigation of raw meal distribution in a model of the calciner placed in FLSmidth R&D Centre Dania. Thereby, a CFD model was built in a way ensuring a geometrical similitude to the calciner used in the experiments with kinematically similar boundary conditions. Numerical simulations were carried out in FLU-ENT.

Some theoretical background was studied to obtain a general idea of gas-solid flows. The most characteristic parameters describing these systems were determined for the flow analyzed in this report. Based on these information, the CFD model was created. The process of building the model involved several aspects. Firstly, a mesh of the calciner was created in GAMBIT. A grid-independency study was carried out in order to choose proper number of cells. Suitable models were chosen for simulate the flow of raw meal particles in the calciner - $k - \varepsilon$ model for continuous phase and DPM model for dispersed phase. A statistically independent solution were found regarding a number of particles tracked in the simulations. Finally, results from the simulations and the experiments were compared and analyzed.

Two cases were considered in present report. Case 1 is the flow of raw meal particles in the calciner with a low ratio of the tertiary air to the total flow ($r_t = 20\%$), whereas case 2 is the flow with a high ratio ($r_t = 55\%$). Results from the simulations for both cases were compared and analyzed regarding flow features, a residence time and a raw meal distribution within several cross-sections of the calciner. Then, these results were compared to the relevant results from the experiments. Two methods were used to compare the results. The first method is based on plotting the temperatures measured in the experiments and the corresponding temperaures resolved from the simulations. The second method is based on plotting the so called weight factor in form of contour plots. The weight fraction corresponds to a fraction of raw meal particles in a certain area of the calciner.

When analyzing the flow features of two types of the flow based on numerical simulations, it seems that case 2, in comparison with case 1 provides better mixing of raw meal particles, and thereby more uniform distribution within the calciner. The reason of such a statement is a high swirling effect, and thereby high turbulence of the flow, especially in lower parts of the calciner. A vortex resulting from the high swirling flow is beneficial to ensure full heat transfer between the air and the particles and good dispersion of the raw meal. However, an analysis of raw meal concentration shows that particles are "better" distributed for case 1. In case

2, the particles flow as a bunch without significant dispersion within the calciner. It means that high swirling effect does not ensure good dispersion. On the other hand, high raw meal concentration was observed in the vicinity of the walls for case 1. This is rather undesirable phenomena leading to a formation of builds-up and thereby a possible damage of full-scale calciner.

A comparison of the simulations and the experiments shows that experimental data for case 1 are in good agreement with the corresponding numerical data. The raw meal distribution is similar along whole calciner. In contrast to case 1, the experimental and the numerical results for case 2 are not comparable. However, one common feature was observed, namely the particles are not spread within the calciner and they move as a one big cluster.

5.1 Future work

The analysis and comparison of the experimental and the numerical results show similarities, expecially for case 1. However, further investigation in this direction has to be made in order to find out more "correct" settings of the CFD model as well as more efficient method of measurements. However, this section only is focused on the numerical part as the main objective of present report. Some of the aspects for the future work are stated and shortly described below.

A more suitable model used for modeling a turbulence needs to be investigated, preferably a RSM model (Reynolds-stress Model). This model, in comparison with the $k - \varepsilon$ model used in present work resolves velocity components in each direction. Thereby, a turbulence is not isotropic and results can be more "correct". However, using this model a computational time and CPU memory would increase significantly. Furthermore, a problem with convergency can occur. Similarly, another multiphase model (for example *Mixture model*) could be investigated.

The simulations described in present thesis are based on a rough estimation of initial conditions of particles injection (velocity components). An investigation of the raw meal injection has to be considered in the future work. This parameter can be a key paramter when analyzing distribution of raw meal particles in the calciner.

This project gives a view of how raw meal particles are distributed within the calciner. This is a good starting point for an analysis of full-scale calciner involving a calcination of raw meal and a combustion of fuel. Thereby, the calciner can be redesigned ensuring more efficient operation.

Bibliography

Alsop, P. A. (2005, January). The Cement Plant Operations Handbook (4th ed.). International Cement Review.

- Arngrímsson, A. H., A. Nielsen, K. T. Borawski, and S. Tan (2008, June). *Experimental investigation of heat transfer* and fluidization in a bed of cement clinker. Student project, Institute of Energy Technology, Aalborg University.
- Bałdyga, J. and J. R. Bourne (1999). Turbulent Mixing and Chemical Reactions. John Wiley & Sons.
- Borawski, K. T. (2008, December). *Raw Meal Suspension In Calciner*. Student project, Institute of Energy Technology, Aalborg University.
- Casey, M. and T. Wintergerste (2000, January). Best Practice Guideline. ERCOFTAC Special Interest Group on "Quality and Trust in Industrial CFD".
- Çengel, Y. A. (2007). Heat and MassTransfer (3rd. ed.). McGraw-Hill.
- Ciborowski, J. (1965). *Podstawy Inżynierii Chemicznej*. Wydawnictwa naukowo-techniczne. (The Basics of Chemical Engineering, written in Polish).
- Clift, R., J. Grace, and M. E. Weber (2005). Bubbles, Drops, and Particles. Dover Publications, Inc.
- Crowe, C., M. Sommerfeld, and Y. Tsuji (1998). Multiphase Flows with Droplets and Particles. CRC Press.
- Crowe, C. T. (2006). Multiphase flow handbook. Taylor & Francis Group.
- Einarsson, Ágúst. Ørn. (2008, February). Process.ppt, internal material. FLSmidth A/S.
- FLUENT (2006, September). FLUENT 6.3 User's Guide. Fluent Inc., Lebanon, NH, USA.
- GAMBIT (2007, May). GAMBIT 2.4 User's Guide. Fluent Inc., Lebanon, NH, USA.
- Haider, A. and O. Levenspiel (1989). Drag coefficient and terminal velocity of spherical and nonspherical particles. *Powder Technology 58*, pp. 63–70.
- Hefny, M. M. and R. Ooka (2009). CFD analysis of pollutant dispersion around buildings: Effect of cell geometry. *Building and Environment 44*, pp. 1699–1706.

Hetsroni, G. (1982). Handbook of MultiPhase Systems. Hemisphere Publishing Corporation.

- Kær, S. K. (2001, October). Numerical investigation of deposit formation straw-fired boilers Using CFD as the framework for slagging and fouling predictions. Ph. D Thesis, Institute of Energy Technology, Aalborg University.
- Kunii, D. and O. Levenspiel (1991). Fluidization Engineering (2nd ed. ed.). Butterworth-Heinemann.
- Maheshwari, S. K. (2009). Stop borrowing from future generations. In Emerging Markets Report, pp. pp. 65–70.
- Mandø, M. (2007). *Turbulence Mixing* presentation from the lecture at 8th semester of the specialization. Fluids and Combustion Engineering, Institute of Energy Technology, Aalborg University.
- Michaelides, E. E. (2005). *Particles, Bubbles & Drops. Their Motion, Heat and Mass Transfer*. World ScientificPublishing Co.

- Portland Cement Association (2008, May). *Economics of the U.S. Cement Industry*. Portland Cement Association. http://www.cement.org/index.asp /Economic Research/Industry Overview. Retrieved in May, 2009.
- Rhodes, M. (2007). Introduction to Particle Technology (2nd ed.). John Wiley & Sons.
- Rosendahl, L. A. (1998, November). *Extending the modelling framework for gas-particle systems*. Ph. D Thesis, Institute of Energy Technology, Aalborg University.
- Rusås, J. (1998, February). *Numerical Simulation of Gas-Particle Flow Linked to Pulverized Coal Combustion*. Ph. D Thesis, Institute of Energy Technology, Aalborg University.
- Shirolkar, J. S., C. F. M. Coimbra, and M. Q. McQuay (1996). Fundamentals aspects of modeling turbulent particle dispersion in dilute flows. *Progress in Energy and Combustion Science 22*, pp. 363–399.

Tennekes, H. and J. L. Lumley (1972). A first course in turbulence. MIT Press, Cambridge, MA.

- United Nations (2007, February). Kyoto Protocol Reference Manual on Accounting of Emissions and Assigned Amounts. Framework Convention On Climate Change. United Nations.
- Versteeg, H. K. and W. Malalasekera (2007). *An Introduction to Computational Fluid Dynamics* (2nd ed.). Pearson Education Limited.



Cement production

(This section is rewritten from the student project mady by Borawski (2008))

Portland cement¹ is manufactured in a series of processes. All of these processes are interconnected and has influence on each other. So it is very important to maintain conditions, for which production is optimum. Figure A.1 shows the typical cement production.

Cement production starts in quarries, where limestone and clay are mined and crushed. Location of the crushers could be at the quarry or at the plant (largely is a function of haulage vs. conveying costs). Already crushed raw materials are mixed in the correct proportions and stored in special stacker/reclaimer systems, where pre-blending is maintained. Afterwards, this raw mixture is ground in a raw mill. The product is a fine powder, called raw meal. Before proceeding further to the process, raw meal is stored in special silos, where it is also well blended. Then it is conveyed to a kiln system.

The basic kiln system comprises a preheater in which raw meal is preheated by heat exchange with hot exhaust gases, a rotary kiln in which the sintering occurs and a cooler in which the hot product from the kiln exchanges heat with the ambient air. Modern kiln systems comprise also a pre-kiln called calciner (secondary combustion vessel between the kiln and the preheater), what makes the process more efficient. There are several types of kiln systems, each with unique advantages depending upon the particular application. The main difference between them is a shape and a functionality of the calciner.

The raw meal is dosed from storage silos to the preheater. The process of preheating is done in a cyclone preheater which consists of one, two or even three parallel strings of cyclones depending on the capacity of the rotary kiln. Each string consists of 4-6 cyclones, which are arranged vertically, one above other in a zigzag formation. Preheating is done as a counter flow process by heat exchange between hot exhaust gases comes from the rotary kiln and the cold raw meal from the silo.

The raw meal, preheated in the cyclone tower is transferred to the calciner, where calcium carbonate dissociates and forms calcium oxide with the evolution of carbon dioxide. This reaction is called calcination and it proceeds at approx. 900°C according to the following formula:

$$CaCO_3 \xrightarrow{heat} CaO + CO_2$$
 (A.1)

¹Portland cement is the most commonly produced cement. There are several types of the cement depending on the composition of a raw mixture and the way of manufacture.

A Cement production

The calcination is strongly endothermic reaction, therefore an extra source of heat is required. The heat is supplied through a combustion of a fuel in a burner, which is placed at the bottom of the calciner. After the calcination, the raw meal is transported to the rotary kiln, where a sintering occurs at approx. 1450°C. To obtain such a high temperature, the burner is placed at the end of the rotary kiln. Basically, the sintering is a reaction of calcium oxide with the other components, and forming calcium silicates and aluminates. But in the reality, the raw meal undergoes a number of complex physicochemical processes.

According to this fact and the practical experience, the raw meal burning process in the kiln can be divided into three main zones: the calcining zone, the burning zone and the cooling zone. In the calcining zone, the final calcination occurs - the calcination degree in the calciner is in the range of 90-95%. During the calcination, a considerable amount of dicalcium silicate is formed ($2CaO \cdot SiO_2$ known as C2S). In the burning zone, firstly the raw meal is heated up to the temperature of approx. $1330^{\circ}C$, when the sintering starts. Temperature still increases and it causes that the liquid phase is formed in the form of nodules - nodulisation. Then, the main component of Portland cement, namely tricalcium silicate (C3S) is obtained according to the following chemical formula:

$$CaO + (2CaO) \bullet SiO_2 \rightarrow (3CaO) \bullet SiO_2$$
 (A.2)

The cooling zone begins, when the material passes the flame coming from the combustion of the fuel in the burner. Temperature decreases and the liquid phase starts to solidify. The final product leaving the rotary kiln (so called clinker) is in the form of black nodular material, with the particle size in the wide range - from 1 to 500 mm. Also a reasonable amount of smaller particles, as well as dust leave the rotary kiln.

The clinker leaves the kiln at approx. 1200°C, and then is cooled down to less than 100°C in the cooler. Usually, process of cooling is done in the grate cooler. The clinker is transported through the cooler by specially designed grate bars and the ambient air is blown up through the moving clinker bed. A part of the air blown through the cooler is used as preheated air for combustion of the fuel in the rotary kiln and the calciner. After the cooling process, the clinker is stored and next milled with a small proportion of gypsum (to control the rate of the hydration). The final product is cement, which is ready for packing and sending to the customers.



Figure A.1: Cement production. Red circle shows the calciner and the preheating tower. The picture is from a presentation made by Einarsson (2008).

В

Scaling gas-solid flows

Systems shows a similitude when adequate dimensionless criteria which are groups of parameters comprised of physical properties describing these system (so called dimensionless numbers) are equal. Basically, there are two theoretical methods used for establishment dimensionless numbers which have to be constant when systems are scaled. First method is used when differential equations describing the process (system) are not known. This method is based on Buckingham π theorem. Second method is applied when these equation are known and application of this method is shown below (based on Hetsroni (1982)).

An equation used for scaling gas-solid flows is the equation of motion of a particle (Equation 2.11) expressed by using the relaxation time:

$$\frac{d\vec{\mathbf{v}}}{dt} = \frac{1}{\tau_p} \left(\vec{u} - \vec{\mathbf{v}} \right) + \vec{g}$$

The particle velocity can be expressed by the displacement vector \vec{x}

$$\vec{\mathbf{v}} = \frac{d\vec{x}}{dt} \tag{B.1}$$

The equation of motion can be rewritten in dimensionless form by introducing a reference velocity U, a reference length D, and the other nondimensional quantities:

$$\vec{x}^* = \frac{\vec{x}}{D} \qquad t^* = \frac{Ut}{D} \qquad \vec{u}^* = \frac{\vec{u}}{U} \qquad \vec{v}^* = \frac{\vec{v}}{U}$$
(B.2)

Hence, the equation of motion becomes

$$\frac{d^{2}\vec{x}^{*}}{dt^{*2}} = \underbrace{\frac{D}{U\tau_{p}}}_{St^{-1}} \left(\vec{u}^{*} - \frac{d\vec{x}^{*}}{dt^{*}} \right) + \underbrace{\frac{Dg}{U^{2}}}_{Fr^{-1}} \vec{I}$$
(B.3)

where \vec{I} is a unit vector in the direction of gravitational acceleration. Thus, for geometrically similar gas-solid dilute flows with kinematically similar boundary conditions, dynamic similarity can be obtained if the following dimensionless numbers are equal for the full-scale system and the model-scale system:

$$St = \frac{U\tau_p}{D}$$
 $Fr = \frac{U^2}{Dg}$ (B.4)

where *St* is already known the Stokes number and *Fr* is the Froude number. As it was shown in Section 2.2.2, the relaxation time and thereby the Stokes number depends on the particle Reynolds number Re_p for non Stokes' flows, which is not known a priori. In order to simplify the dimensional analysis, the particle Reynolds number can be expressed as

$$Re_{p} = \frac{d_{p}|\vec{u} - \vec{v}|\rho}{\mu} = Re_{p}^{*} \left| \vec{u}^{*} - \frac{d\vec{x}^{*}}{dt^{*}} \right|$$
(B.5)

where Re_p^* is so called the pseudo-particle Reynolds number described as the following

$$Re_p^* = \frac{d_p U \rho}{\mu} \tag{B.6}$$

Thus, instead of complex form of the Stokes number (non Stokes' flows), the pseudoparticle Reynolds number and the Stokes number for Stokes' flows can be used. Concluding the analysis presented above, scaling dilute flows requires consideration of the three dimensionless parameters: the Stokes number (for Stokes' flow), the pseudo-particle Reynolds number and the Froude number. It has to be mentioned, that there are more criteria which has to be fulfilled. These criteria are: the volume fraction (Equation 2.3.1) and the Reynolds number calculated for the main flow. However, the latter causes so called distortion problem, because it is impossible to keep constant simultaneously the Reynolds number and the Froude number (unless the fluid with different properties would be used).

C

The Navier-Stokes equations

The Navier-Stokes equations (Equation C.5) are the basic differential equations describing the flow of Newtonian fluids¹. The N-S equations are comprised of the equation of motion - Equation C.1 and the conservation of mass equation, so called continuity equation - Equation C.4. The equation of motion is based on Newton's Second Law and it describes motion of an element as a result of body and surface forces accting on this element. The equations of motion in the *x*, *y*, and *z* direction are as follows:

(x direction)
$$\rho g_x + \frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} = \rho \left(\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right)$$
 (C.1)

(y direction)
$$\rho g_y + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} = \rho \left(\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right)$$

(z direction)
$$\rho g_z + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_{zz}}{\partial z} = \rho \left(\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right)$$

where u, v, and w are the velocity components in the direction x, y, and z, respectively; g_x , g_y and g_z are components of the gravitational acceleration. Symbols σ and τ represent normal stress and shearing stresses, respectively. For Newtonian fluid, the normal and the shear stresses are expressed by the following equations:

$$\sigma_{xx} = -p + 2\mu \frac{\partial u}{\partial x} \qquad \sigma_{yy} = -p + 2\mu \frac{\partial v}{\partial y} \qquad \sigma_{zz} = -p + 2\mu \frac{\partial w}{\partial z} \tag{C.2}$$

$$\tau_{xy} = \tau_{yx} = \mu \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \qquad \tau_{yz} = \tau_{zy} = \mu \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right) \qquad \tau_{zx} = \tau_{xz} = \mu \left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \right) \tag{C.3}$$

The continuity equation is based on the principle which says that the mass of a system remains constant as the system moves through the flow field. For steady flow of incompressible fluids ($\rho = \text{const}$), the continuity equation is described as:

¹Newtonian fluids are characterized by linear relation of the shearing stress to the rate of shearing strain, which is $\tau \propto \frac{du}{dx}$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{C.4}$$

Combining and rearranging equations of motion and the continuity equation yield the Navier-Stokes equations:

(x direction)
$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = -\frac{\partial p}{\partial x} + \rho g_x + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \quad (C.5)$$

$$(y \quad \text{direction}) \qquad \rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = -\frac{\partial p}{\partial y} + \rho g_y + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right)$$

(z direction)
$$\rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) = -\frac{\partial p}{\partial z} + \rho g_z + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right)$$

In the equation above, terms on the left side represent acceleration of the element, terms on the right describe the forces acting on this element.

C.1 How to solve Navier-Stokes equations?

Over the years, many methods describing the motion of the fluid have been developed, but three of them are the most commonly used due to their correctness. These methods are: RANS, LES and DNS. RANS (Reynolds-averaged Navier-Stokes equations) is described in the main part of the project - see Section 2.1. LES and DNS are described shortly below.

Direct Numerical Simulations (DNS)

Basically, DNS are the three-dimensional, time-dependent numerical solutions of the Navier-Stokes equations. Despite the fact, that computer technology is becoming more and more advanced, DNS due to their high computer requirements are not used for practical applications - they are limited to relatively low Reynolds numbers (lower than 10.000 (Mandø, 2007)). This method requires huge amount of meshpoints at which the NS equations are being solved. An estimation of the number of meshpoints *N* required is shown in Equation C.6 (Bałdyga and Bourne, 1999). Furthermore, the time step has to be so small that an element of the fluid will not move further than one meshlength during the time step. The total computation time t_{comp} can be estimated by Equation C.7 (Bałdyga and Bourne, 1999).

$$N \simeq \left(\frac{L}{\eta}\right)^3 \approx R e_L^{9/4} \tag{C.6}$$

$$t_{comp} \sim N^{4/3} \sim Re_L^{-3} \tag{C.7}$$

where η is the size of the smallest eddy; *L* is the characteristic dimension of the system; Re_L is the Reynolds number. As an example, the flow with Re = 10000 will be analysed. According to the formulas above, the number of meshpoints required to solve the velocity field is approximately 10 millions, hence the computation time must be presented in years.

Large Eddy Simulations (LES)

LES methods are very similar to DNS and also based on solving Navier-Stokes equations. They have been developed to decrease the number of meshpoints by using more coarse grid and employing numerical model of turbulence for smaller scale (*e.g.* Smagorinsky's model). Thanks to that fact, three-dimensional, time-dependent flow field can be obtained for high Reynolds numbers. Although, the number of meshpoints as well as computation time are much smaller comparing to DNS, LES methods are still too demanding regarding computer storage and CPU time. Therefore, LES is not commonly used in industrial applications. Although this trend is expected to change in the future.

Comparison of velocity profiles for grids with various number of cells



Figure D.1: Comparison of the velocity profiles for grids with different number of cells. Velocity profiles in the "swan neck" before the bend (left plot) and after the bend (right plot), vertical and normal to the main direction of the flow.



Figure D.2: Comparison of the velocity profiles for grids with different number of cells. The left plot: velocity profile in the "swan neck" at z = 3.7 (m) along the *x* axis, the right plot: velocity profile in the "swan neck" along the *y* axis.

D Comparison of velocity profiles for grids with various number of cells



Figure D.3: Comparison of the velocity profiles for grids with different number of cells. Velocity profile at the bend of the "swan neck", horizontal and normal to the main direction of the flow.

E

Particle size distribution

Size distribution of the raw meal used in experiments described in Borawski (2008) was measured by a device called *MasterSizer 2000* which is based on laser diffraction technology. Basically, light from a laser is shone into a cloud of particles suspended in a transparent gas *e.g.* air. The particles scatter the light, but smaller particles scatter the light at larger angles than bigger particles. The scattered light is measured by photodetectors placed at different angles and this is known as the diffraction pattern. By using light scattering theory (Mie theory or Lorenz-Mie theory) which is basically an analytical solution of Maxwell's equations for the scattering of electromagnetic radiation, particle size distribution can be estimated.

There are assumptions made in the light scattering theory:

· Particles are assumed to be spherical

Diameter is estimated from the volume of "spherical" particle measured and calculated by analyzer.

• the suspension is dilute

Scattered light is directly measured by detectors, it is assumed that the light is not rescattered by other particles before reaching the detector.

• Homogeneous phase

If a mixture of different materials is analyzed, it is assumed that light is scattered in the same way for all particles, although each component has different characteristic of scattering the light.

The scan of the report from the measurements of PSD for raw meal before and after the experiments is presented on the next pages.



Operator notes: Average of 3 measurements from Project - Raw meal 2, 2975 rpm, 60 s. us.

Figure E.1: Particle size distribution of raw meal before the experiments. Measurements were carried out in FLSmidth R&D Centre Dania.





Result Analysis Report



Operator notes: Average of 3 measurements Project - Raw meal, 2975 rpm, 60 s. us.

Figure E.2: Particle size distribution of raw meal after experiments. Measurements were carried out in FLSmidth R&D Centre Dania.

Particle concentration profiles for simulations with various number of particles



Figure F.1: Comparison of the particle concentration profiles for simulations with different number of particles. The left plot shows the profile at z = 2.895 (m) in the *x* direction, whereas the right plot in the *y* direction.



Figure F.2: Comparison of the particle concentration profiles for simulations with different number of particles. The left plot shows the profile at z = 3.195 (m) in the *x* direction, whereas the right plot in the *y* direction.



F Particle concentration profiles for simulations with various number of particles

0.2

Position (m)

0

-0.4

-0.2



0.4

0

-0.4

-0.2

0.2

0

Position (m)

0.4



Figure F.4: Comparison of the particle concentration profiles for simulations with different number of particles. The left plot shows the profile at the outlet in the x direction, whereas the right plot in the y direction.



Figure F.5: Comparison of the particle concentration profiles for simulations with different number of particles. The plot shows the profile in the bend of the swan neck in the z direction.

Comparison of temperatures profiles for simulations and experiments



Figure G.1: Comparison of the temperature profiles for the experiments and the simulations. Data correspond to the lines A and B for case 1 at z = 2.895 (m).



Figure G.2: Comparison of the temperature profiles for the experiments and the simulations. Data correspond to lines C and D for case 1 at z = 2.895 (m).

Relevant parameters describing the flow of raw meal particles in the calciner were determined based on theoretical background. Based on this information, an appropriate turbulence model and multiphase model were chosen. A description of building the model involves several aspects *e.g.* a mesh generation and grid-independence study, an analysis of a sufficient number of particles tracked in the simulations and relevant calculations required





Figure G.3: Comparison of the temperature profiles for the experiments and the simulations. Data correspond to the lines A and B for case 1 at z = 3.195 (m).



Figure G.4: Comparison of the temperature profiles for the experiments and the simulations. Data correspond to lines C and D for case 1 at z = 3.195 (m).

for determination of boundary conditions.

Two extreme cases were considered in present work: the flow with strong and weak swirling effect. Thereby, the influence of revolving flow on raw meal distribution was investigated. Finally, the results from the simulations were compared and analyzed with the experimental data. This analysis shows that



Figure G.5: Comparison of the temperature profiles for the experiments and the simulations. Data correspond to the lines A and B for case 1 at z = 3.495 (m).



Figure G.6: Comparison of the temperature profiles for the experiments and the simulations. Data correspond to lines C and D for case 1 at z = 3.495 (m).



Figure G.7: Comparison of the temperature profiles for the experiments and the simulations. Data correspond to the lines A and B for case 2 at z = 2.895 (m).





Figure G.8: Comparison of the temperature profiles for the experiments and the simulations. Data correspond to lines C and D for case 2 at z = 2.895 (m).



Figure G.9: Comparison of the temperature profiles for the experiments and the simulations. Data correspond to the lines A and B for case 2 at z = 3.195 (m).



Figure G.10: Comparison of the temperature profiles for the experiments and the simulations. Data correspond to lines C and D for case 2 at z = 3.195 (m).



Figure G.11: Comparison of the temperature profiles for the experiments and the simulations. Data correspond to the lines A and B for case 2 at z = 3.495 (m).



Figure G.12: Comparison of the temperature profiles for the experiments and the simulations. Data correspond to lines C and D for case 2 at z = 3.495 (m).

G Comparison of temperatures profiles for simulations and experiments