STELLINGEN

Behorende bij het proefschrift

On the sedimentation process in a trailing suction hopper dredger

van

Cees van Rhee

Delft, 3 december 2002

1. Indien voor de bezinking van sediment in het beun van een sleeppopperzuiger een eenvoudig bezinkmodel wordt gekozen, kan beter gebruik worden gemaakt van een model met een verticale doorstroming, dan de gebruikelijke modellen met een horizontale stroomrichting.

2. Algemeen wordt aangenomen dat de stroomsnelheid nabij het zandbed in het beun toeneemt tijdens het laden ten gevolge van een afnemende mengseldiepte. Het tegenovergestelde is echter het geval: afgezien van de slotfase is deze snelheid bij aanvang laden het hoogst, waarna deze geleidelijk in de tijd afneemt.

3. In tegenstelling tot wat meestal wordt aangenomen heeft de toenemende sediment concentratie in overvloei tijdens het laden niets met afname van de mengseldiepte (en de daarbij veronderstelde toename van de erosie) te maken.

4. In de meeste morfologische modellen is de relatie tussen de sedimentatiesnelheid en de concentratie vlak boven het bed fout geïmplementeerd omdat de sedimentatiesnelheid ten opzichte van de valsnelheid wordt verwaarloosd. Voor lage concentratie is dit niet zo erg, maar voor hoge concentratie (sheetflow omstandigheden) leidt dit tot een zeer grote fout in de voorspelling van de sedimentatiesnelheid. (zie par. 3.11 van dit proefschrift)

5. Door de optredende schaaleffecten kunnen de resultaten van model beunbezink proeven niet zondermeer naar prototype omstandigheden worden verschraald.

6. Ten opzichte van de grote hoeveelheid informatie over de discretisatie van vergelijkingen in het interne gedeelte van het berekeningsdomein is slechts weinig over de discretisatie van de randen gepubliceerd. Dit is vreemd aangezien de stroming voor het grootste deel door de randvoorwaarden wordt bepaald.
7. Bij de meeste markstudies leiden de aangenomen omstandigheden meestal tot een bevestiging van een vooraf ingenomen standpunt.

8. Bij een brand in Nederland schijnen nooit schadelijke stoffen vrij te komen. In dat licht bezien is het merkwaardig dat bij vuilverbrandingsovens zoveel moeite wordt gedaan om de rookgassen te zuiveren.

9. De overeenkomst tussen wiskundigen en juristen is dat zij eenvoudige materie, door gebruik te maken van ingewikkelde formuleringen, toch zeer gecompliceerd weten te maken.

10. De flinke toename van het aantal voorrangskruisingen, nadat ook wielrijders van rechts voorrang kregen, is een mooi voorbeeld van hoe een stompzinnige regel toch nog positief kan uitwerken.

11. De stelling "ieder nadeel heb z’n voordeel" is ook van toepassing op de kwaliteit van Nederlandse TV programma’s en het promoveren in eigen tijd.

12. Groene stroom is windhandel.

Deze stellingen worden verdedigbaar geacht en zijn als zodanig goedgekeurd door de promotoren,

Prof. Ir. W.J. Vlasblom

Prof. Ir. K. d’Angremon
On the sedimentation process in a trailing suction hopper dredger

Over het sedimentatie proces in een sleeppopperzuiger

TN 39 66
On the sedimentation process in a trailing suction hopper dredger

Proefschrift

ter verkrijging van de graad van doctor aan de Technische Universiteit Delft, op gezag van de Rector Magnificus prof. dr. ir. J. T. Fokkema, voorzitter van het College van Promoties, in het openbaar te verdedigen op dinsdag 3 december 2002 te 16:00 uur

door Cornelis Van Rhee
civiel ingenieur
geboren te Leiden
Dit proefschrift is goedgekeurd door de promotoren:
Prof. ir. W.J. Vlasblom
Prof. ir. K. d'Angremond

Samenstelling Promotiecommissie:

Rector Magnificus, voorzitter
Prof. ir. W.J. Vlasblom                     Technische Universiteit Delft, promotor
Prof. ir. K. d'Angremond                   Technische Universiteit Delft, promotor
Dr. ir. J.C. Winterwerp                    Technische Universiteit Delft, toeg. promotor
Prof. dr. ir. J.E. Berlamont               Katholieke Universiteit Leuven (België)
Prof. dr. ir. F.T.M. Niewstadt             Technische Universiteit Delft
Prof. dr. ir. J.F. Richardson              University of Wales Swansea (UK)
Prof. dr. ir. L.C. van Rijn                Universiteit Utrecht

The studies described in this thesis have been financed by the Vereniging van waterbouwers in Bagger-, Kust- en Oeverwerken (VBKO).

Copyright © 2002 by Cornelis van Rhee
Printed by printpartners Ipskamp.
To Monique
Abstract

The Trailing Suction Hopper Dredge (TSHD) was originally developed for maintenance dredging. However, during the last 10 years the TSHD (or hopper dredger) has become more involved in Capital Dredging. The large land reclamation projects in Hong Kong and Singapore were and are being realised by a large fleet of hopper dredgers.

During dredging the TSHD lowers one or two suction pipes to the seabed. From the bed a sand-water mixture is sucked up and discharged into a large cargo hold, the so-called hopper. Provided the sediment in the mixture is not too fine a separation process takes place in the hopper under gravity. The sediment settles in the hopper and the excess water flows overboard. A part of the inflowing sand may not settle during loading into the hopper, but flows back overboard with the excess water. Depending on the particle size distribution (PSD) of the sediment, the hopper geometry and other process parameters this overflow loss can reach values up to 30-40% of total volume dredged. It is important to quantify these losses both for the quality (PSD of sand loaded) and quantity (loading production).

Therefore, the VBKO (Vereniging van waterbouwers in Bagger-, Kust- en Oeverwerken) initiated a research program to improve knowledge on the subject of hopper sedimentation.

In the past different models were developed to estimate the amount of material lost overboard, the particle size distribution (PSD) of the sediment in the hopper and the PSD in the overflow mixture. These models were based on relative simple models developed for the field of sewage water treatment (Camp, 1946) and all have idealised or simplified inflow and outflow configurations and a prescribed velocity distribution in the hopper. The objective of the current investigation was to develop a new model to simulate the hopper sedimentation process because the existing models have too many restrictions.

The research program started in 1997 and can be divided into three parts: Laboratory experiments; numerical modelling; and prototype validation of the models. The laboratory experiments can be divided into model hopper sedimentation tests and more fundamental tests on one-dimensional sedimentation and the influence of bottom shear stress and air bubbles on sedimentation.

The research program started with a literature survey and formulation of scaling rules for laboratory tests. Hereafter laboratory tests were executed with large dimensions to minimise scale effects. During these tests the in- and outflow quantities were monitored and much effort was concentrated on taking measurements inside the hopper (velocity and concentration) to get a better understanding of the physics involved. This intensive test program resulted in a better perception of the flowfield and sedimentation process in the hopper.
The laboratory test revealed that buoyancy effects dominated the flow inside the hopper. The flow velocity close to the settled bed was relative high due to the presence of a density current. In the remaining part of the hopper the flow was in a vertical direction rather than horizontal. This finding motivated the development of a one-dimensional model in vertical direction. In the 1DV model the supply of the sediment-water mixture is simplified. The mixture is equally distributed over the length and width of the hopper. The model includes the important effect of the vertical velocity component and the mutual interaction of the different grain sizes of the particle size distribution. This latter effect is totally absent in the above-mentioned simple models for the sewage water treatment, where every fraction is calculated independently. The 1DV model was validated using the results of one-dimensional sedimentation tests and proved also capable to simulate the (three-dimensional) laboratory hopper sedimentation tests.

It was however uncertain if the model would simulate the process on prototype scale equally well since the horizontal transport in the hopper was not included owing to the one-dimensional character. It was therefore decided to extend the one-dimensional model to two dimensions. The hydrodynamic 2DV model developed is based on the Reynolds Averaged Navier-Stokes equations with a k-epsilon turbulence model. The model includes the influence of the overflow level of the hopper (moving water surface) and a moving sand bed due to the filling of the hopper. The sediment transport equations are coupled to the momentum equations to enable the development of buoyancy-driven flows. The influence of the particle size distribution (PSD) is included in the sediment transport equations.

A boundary condition at the interface between the settled sediment and the mixture above must be formulated for the numerical model. It was recognised that quantitative information on the influence of the bed shear stress on sedimentation was missing for the situation present in a hopper (large concentration and relative low flow velocities). Therefore unique sedimentation tests were carried out in closed flumes. In the first instance pilot tests were carried out at WL|Delft Hydraulics. Later these tests were followed by a test program in the Dredging Technology Research Laboratory of the Delft University of Technology with larger dimensions and extensive instrumentation. An empirical relation based on these experiments was built in the two-dimensional model. The model was validated using laboratory sedimentation tests and prototype measurements. The agreement between the measurements and model is encouraging.
Samenvatting

De sleephopperzuiger werd oorspronkelijk voornamelijk gebruikt voor onderhoudsbaggerwerk, zoals bijvoorbeeld het op diepte houden van vaarwegen. Gedurende de laatste tientallen jaren werden deze baggerschepen steeds meer ingezet voor ‘Capital Dredging’. De grote landaanwinningswerken in Hong Kong en Singapore zijn hoofdzakelijk met behulp van sleephopperzuigers gerealiseerd.

Tijdens het baggeren zorgt dit type baggerwerktuig een sediment-water mengsel op van de zeebodem, gebruikmakende van één of twee zuigbuizen. Dit mengsel wordt in het laadruim (ook wel het buit genoemd) gestort. Indien het sediment niet te fijnkorrelig is, zal onder invloed van de zwaartekracht een scheidingsproces optreden: het sediment bezinkt en het overtollige water stroomt overboord. Afhankelijk van o.a. de korrelverdeling van het sediment zal een deel van het sediment met het overtollige water overboord spoelen. Deze verliezen kunnen oplopen tot ca. 30 – 40 % van de totale gebaggerde hoeveelheid. In het belang van de productie van het schip en de sedimentkwaliteit is het van belang de grootte van deze zogenaamde overvloeiveliezen te kennen. Daarnaast kan deze kennis van belang zijn in verband met milieueisen, die een limiet kunnen stellen aan de hoeveelheid sediment welke door de baggeractiviteiten in suspensie wordt gebracht.

Ten einde de kennis met betrekking tot het beladingsproces in een sleephopperzuiger te vergroten, is een onderzoeksprogramma geïnitieerd door de Vereniging van waterbouwers in Bagger-, Kust- en Oeverwerken (VBKO). Het uiteindelijke doel van het onderzoek was het ontwikkelen van een theorie of model waarmee het proces gesimuleerd kan worden.

In het verleden zijn modellen ontwikkeld om het overvloeiverlies, de korrelverdeling in de overvloei en de korrelverdeling van het sediment bezonken in de hopper, te bepalen. Deze modellen zijn gebaseerd op relatief eenvoudige modellen welke oorspronkelijk voor de (riool)waterzuivering waren ontwikkeld (Camp, 1946). Deze modellen hebben vereenvoudigde of geïdealiseerde instroom en uitstroom configuraties. Tevens is de stroomsnelheidsverdeling opgelegd en niet gekoppeld aan de sediment concentratie verdeling.

Het onderzoek is gestart in 1997 en bestaat uit drie delen: laboratorium proeven, numerieke modellering en prototype validatie van de modellen. Het experimentele gedeelte kan worden onderverdeeld in model hopper sedimentatieproeven, één-dimensionale sedimentatie proeven in een sedimentatiekolom, sedimentatieproeven in een gesloten goot om de invloed van de bodemschuifspanning op de sedimentatie te onderzoeken en sedimentatie van zand in een bellenkolom teneinde de invloed van luchtbellen op de valsnelheid van zandkorrels te bepalen.
Een literatuuronderzoek en een studie naar schaalregels is uitgevoerd als voorbereiding op grootschalige model hopper sedimentatieproeven. Tijdens deze proeven is, naast de gebruikelijke metingen aan de in- en uitstroming van het beun, zeer veel aandacht besteed aan het meten van snelheids- en concentratie verdelingen in het beun. Met name deze laatste metingen hebben het inzicht in de fysica sterk verbeterd. Uit het proevenprogramma bleek dat de stroming in het beun door dichtheids effecten wordt gedomineerd. De snelheid vlak bij het bezonken zandbed is relatief groot door de aanwezigheid van een dichtheidsstroming. In het overige deel van het beun is de stroming echter niet horizontaal, maar juist verticaal gericht. Dit gaf aanleiding tot de ontwikkeling van een één-dimensionaal verticaal (1DV) sedimentatiemodel. De toever van sediment in dit 1DV model is geschematiseerd. Het instromende mengsel wordt gelijkmatig over het totale oppervlak verdeeld. In het model zit wel het belangrijke effect van de verticale doorstroming en de wederzijdse beïnvloeding van de verschillende korrelfracties tijdens het bezinken. Dit laatste effect is afwezig in de bestaande modellen waar de verschillende fracties onafhankelijk van elkaar worden berekend. Met het 1DV model kunnen de resultaten van de één-dimensionale bezinkproeven en ook de model hopper sedimentatieproeven zeer goed gesimuleerd worden.

De goede resultaten van het 1DV model op modellschaal gaf echter nog geen garantie voor een goede voorspelling op prototype schaal, omdat het horizontale transport van sediment niet in het model opgenomen is. Dit horizontaal transport gaat gepaard met een relatief grote stroom snelheid (dichtheidsstroming) waardoor de sedimentatie kan worden verminderd.

Daarom is besloten een twee-dimensionaal verticaal (2DV) model te ontwikkelen waarin wel dit horizontale transport is opgenomen en tevens ook de invloed van instroom- en overvloeicijfers beter kan worden gemodelleerd. Het 2DV model is gebaseerd op de Reynolds-gemiddelde Navier-Stokes vergelijkingen met een $k-\varepsilon$ turbulentie model. In het model zijn de sediment transportvergelijkingen gekoppeld aan de impulsvergelijkingen, zodat dichtheids aangedreven stromingen goed worden weergegeven. In het model is tevens de variatie van het wateroppervlak en bodemligging (ten gevolge van sedimentatie) opgenomen. De invloed van de korrelderiving is verdisconteerd door de sediment transportvergelijkingen voor meerdere fracties (onderling gekoppeld) op te lossen.

Een belangrijk aspect in het 2DV model is de erosie-sedimentatie randvoorwaarde tussen de stroming en het bezonken zandbed. Het zand sedimenteert in het beun bij een combinatie van een hoge concentratie en een lage bodemschuifspanning. Omdat bij deze unieke situatie weinig kwantitatieve informatie voorhanden is, zijn speciale sedimentatie proeven uitgevoerd in een gesloten goot. Met deze proeven kon op kleinere schaal de situatie vlak bij het bed in het beun van een hopper worden gesimuleerd. Allereerst werden pilot tests uitgevoerd bij WL|Delft Hydraulics. Hierna x
werden proeven uitgevoerd in het Laboratorium van Grondverzet en Bulktransport van de Technische Universiteit Delft. Op grond van de proefresultaten is een empirische relatie opgesteld welke in het 2DV model wordt toegepast. Het 2DV model is gevalideerd met de model hopper sedimentatie testen en prototype metingen. Het 2DV model blijkt het proces op prototype schaal beter te beschrijven dan het 1DV model. De overeenkomst tussen het model en de metingen is redelijk goed.
Contents

ABSTRACT .......................................................................................................................... VII

SAMENVATTING ........................................................................................................... IX

1 INTRODUCTION ........................................................................................................... 1
  1.1 GENERAL .................................................................................................................. 1
  1.2 AIM OF THIS STUDY ............................................................................................... 2
  1.3 RESEARCH METHODOLOGY AND SET UP OF THESIS .................................. 3

2 SEDIMENT MOVEMENTS IN A TRAILING SUCTION HOOPER DREDGER .......................... 7
  2.1 INTRODUCTION ...................................................................................................... 7
  2.2 INTRODUCTION HOOPER DREDGER AND DREDGING PROCESS ...................... 7
    2.2.1 Dimensions of Trailing Suction Hopper Dredgers ............................................ 9
  2.3 EXISTING MODELS .............................................................................................. 10
    2.3.1 Influence of turbulence ..................................................................................... 12
    2.3.2 Influence of turbulence according to Camp and Dobbins ............................... 13
    2.3.3 Vlasblom and Miedema .................................................................................... 15
    2.3.4 Implementation of erosion according to Van Rhee .......................................... 17
  2.4 MODELLING OF THE DECREASING FLOW DEPTH IN THE SETTLING ZONE ........ 18
    2.4.1 Dynamic extension of Ooijens ............................................................................ 19
  2.5 YAGI'S MODEL ...................................................................................................... 19
    2.5.1 Discussion ......................................................................................................... 22
    2.5.2 Average loading efficiency .............................................................................. 23
  2.6 SHORTCOMINGS OF THE DIFFERENT SIMPLE MODELS FOR THE APPLICATION IN
    DREDGING INDUSTRY ............................................................................................... 24
  2.7 HYDRODYNAMIC MODELS ................................................................................... 24

3 SETTLING VELOCITY OF SEDIMENTS ...................................................................... 27
  3.1 INTRODUCTION ...................................................................................................... 27
  3.2 SETTLING VELOCITY OF A SINGLE GRAIN ...................................................... 27
  3.3 PARTICLE RESPONSE TIME – STOKES NUMBER ............................................. 29
  3.4 INFLUENCE OF THE CONCENTRATION ............................................................. 30
  3.5 THE INFLUENCE OF THE PARTICLE SIZE DISTRIBUTION ............................... 32
  3.6 INFLUENCE OF CLAY FRACTION .......................................................................... 35
    3.6.1 Introduction ....................................................................................................... 35
    3.6.2 Settling in a stagnant clay suspension ............................................................. 36
    3.6.3 Setting in shear flow ......................................................................................... 37
7 TWO DIMENSIONAL SEDIMENTATION MODEL (2DV)..........................123

7.1 INTRODUCTION...........................................................................123

7.1.1 Summary of the process conditions to be included in the model...........123

7.2 LIMITATIONS OF THE MODEL .....................................................125

7.3 BASIC EQUATIONS .......................................................................125

7.3.1 Motivation of the mixture model ..................................................125

7.3.2 Conservation of Mass and Momentum .........................................126

7.3.3 Turbulent Flow............................................................................129

7.3.4 Eddy viscosity concept.................................................................130

7.3.5 Turbulent closure with the $k$-$\epsilon$ model.................................132

7.3.6 The Re-Normalisation Group (RNG)- $k$-$\epsilon$ model.....................133

7.4 SEDIMENT TRANSPORT.................................................................134

7.4.1 Coupling of the fractions ............................................................136

7.4.2 Influence of grain-grain interaction .............................................137

7.5 BOUNDARY CONDITIONS..............................................................139

7.5.1 Horizontal and vertical velocity ....................................................139

7.5.2 Boundary conditions for the $k$-$\epsilon$ model.................................142

7.5.3 Boundary Conditions for the Sediment Transport equations ..........144

7.6 OVERVIEW OF THE MODEL ........................................................146

7.7 OVERVIEW NUMERICAL METHODS ..............................................147

7.7.1 Introduction ................................................................................147

7.7.2 Overview of numerical methods to solve partial differential equations ..148

7.7.3 Cartesian versus Boundary fitted grids ........................................149

7.8 NUMERICAL IMPLEMENTATION SEDIMENT TRANSPORT EQUATIONS ...149

7.8.1 Advection Diffusion Equation in Conservative Form .....................150

7.8.2 Interpolation Practices.................................................................152

7.8.3 Approximation of the diffusive flux ..............................................153

7.9 NUMERICAL IMPLEMENTATION OF THE NAVIER-STOKES EQUATIONS ..155

7.9.1 Introduction ................................................................................155

7.9.2 Basic problem with NS-equation for an incompressible fluid ...........155

7.9.3 Possible Solutions .......................................................................156

7.9.4 Turbulent Shear Stresses ............................................................157

7.9.5 Advection ...................................................................................158

7.9.6 Pressure gradient ......................................................................159

7.10 NUMERICAL IMPLEMENTATION OF THE $k$-$\epsilon$ MODEL..................161

7.11 TREATMENT OF THE BOUNDARIES .............................................165

7.12 APPLICATION OF THE 2DV MODEL .............................................168

7.12.1 Introduction ...............................................................................168

7.12.2 Channel flow clear water ...........................................................168
7.12.3 Backward facing step................................................................. 169
7.12.4 Buoyant and non-buoyant plane jets ........................................... 171
7.12.5 Sediment transport in a rectangular channel................................. 176
7.12.6 Hyperconcentrated flow ........................................................... 177
7.12.7 Concluding remarks hyperconcentrated flow................................ 182
7.12.8 Buoyant plume on a sloping bed ............................................... 183
7.12.9 Concluding remarks test calculations ........................................ 184
7.13 APPLICATION OF THE 2DV MODEL FOR HOPPER SEDIMENTATION .... 185
7.13.1 Introduction .............................................................................. 185
7.13.2 General settings of the 2DV calculations ..................................... 185
7.13.3 Influence of the grid size .......................................................... 186
7.13.4 Influence of the inflow velocity ................................................... 187
7.13.5 Model hopper sedimentation tests .............................................. 189
7.13.6 Scaling of the results to prototype values and comparison with 1DV model 199
7.14 CONCLUDING REMARKS TESTING THE 2DV MODEL ....................... 202

8 SIMULATION OF PROTOTYPE MEASUREMENTS TAKEN ON BOARD
THE TSHD “CORNELIA”........................................................................... 203
8.1 INTRODUCTION ............................................................................ 203
8.2 HOPPER DIMENSIONS .................................................................... 204
8.3 OPERATIONAL PARAMETERS .......................................................... 204
8.4 MEASUREMENTS .......................................................................... 204
8.5 OVERFLOW LOSS BASED ON HOPPER LOAD PARAMETERS .......... 205
8.6 COMPARISON WITH 1DV MODEL .................................................... 205
8.7 RESULTS OF THE 2DV MODEL ....................................................... 206
8.7.1 Measured and computed overflow losses ..................................... 206
8.7.2 Particle size distribution in overflow ......................................... 208
8.7.3 Velocity and concentration profiles .......................................... 208
8.7.4 Segregation in the hopper ......................................................... 211
8.8 CONCLUDING REMARKS 2D MODEL ............................................. 212

9 CONCLUSIONS AND RECOMMENDATIONS .................................. 213
9.1 INTRODUCTION ............................................................................ 213
9.2 PHYSICAL PROCESSES INSIDE THE HOPPER ............................ 213
9.3 HOPPER SEDIMENTATION TESTS ............................................... 214
9.4 CLOSED FLUME SEDIMENTATION TESTS ..................................... 214
9.5 MODELLING OF THE HOPPER SEDIMENTATION PROCESS ............ 215
9.5.1 Existing models ........................................................................ 215
9.5.2 One-dimensional vertical (1DV) hopper sedimentation model ....... 215
9.5.3 Two-dimensional vertical (2DV) hopper sedimentation model.................216
9.6 RECOMMENDATIONS...............................................................................217

REFERENCES.................................................................................................219

APPENDIX A - LIST OF SYMBOLS.................................................................227

APPENDIX B - ONE DIMENSIONAL SEDIMENTATION TESTS ..................233

APPENDIX C – GRID TRANSFORMATION.......................................................235

APPENDIX D – INFLUENCE OF SPECIFIC DENSITY CORRECTION FOR UNIFORM SEDIMENT........................................................................239

ACKNOWLEDGEMENTS..................................................................................241

CURRICULUM VITAE......................................................................................243
Chapter 1
Introduction

1.1 General
One of the most important types of dredging equipment nowadays is the Trailing Suction Hopper Dredger (TSHD). Originally this type of dredger was developed for maintenance dredging, for instance maintaining the depth of entrance channels of ports. During the last decades however the TSHD has evolved towards Capital Dredging. The large land reclamation projects in Hong Kong and Singapore were and are being realised by this type of equipment. A TSHD is a ship that is equipped with one or two suction pipes, which are lowered to the seabed during dredging. From the seabed a sediment-water mixture is sucked up and discharged into a large cargo hold, the so-called hopper. Provided the sediment in the mixture is not too fine a separation process takes place in the hopper under gravity. The sediment settles in the hopper and the excess water flows overboard. The dredging process is continued until the ship is loaded to her capacity. Thereafter the ship sails to the discharge location where the sand is dumped by opening doors in the bottom or pumped ashore using the dredge pumps of the ship.

Generally a part of the incoming sediment does not settle in the hopper during dredging, but flows overboard with the excess water. Depending on the particle size distribution (PSD) of the sediment, the hopper geometry and other process parameters this overflow loss can reach values up to 30-40% of the total volume of sediment pumped into the hopper. It is important to quantify the overflow loss for three reasons:
- The loading time, and therefore the cost price of the sediment dredged, increases due to the overflow losses.
- In particular the finer fractions of the particle size distribution (PSD) are present in the overflow mixture, implying that the PSD of the settled sediment in the hopper is different from the natural PSD in the seabed. It is important to quantify this effect since the characteristics of sediment to be delivered are often specified under the contract.
- For environmental reasons it is important to quantify both the quantity as well as the PSD of the overflow mixture since these values are (amongst others) needed in the models to estimate the dispersion of the sediment flowing overboard.
The volume of sand dredged and transported by trailing suction hopper dredgers increased spectacularly in the last ten years. To accommodate this growth in total production the world hopper fleet expanded. Both the number of hopper dredgers and the size of these ships increased dramatically. Owing to the increasing importance of the hopper dredgers the need for better methods to describe the hopper sedimentation process grew likewise.

1.2 Aim of this study

We can distinguish three types of sediment considering hopper sedimentation:

- Very fine grained sediments such as clay and silt. The settling velocity of these sediments is low and no settling takes place from the mixture pumped in the hopper. This type of sediment is loaded until the mixture surface reaches the overflow level (Figure 2.4). Hereafter the hopper sails to the discharge location.

- Coarse sands and gravel. The settling velocity of these sediments is large and all sediment pumped into the hopper settles. Overflow losses will be very low.

- Sediment with a particle size distribution situated in between these two above-mentioned classes. The largest part of the sediment discharged in the hopper settles, but a significant part flows overboard.

The flow of all above mentioned sediment types in a hopper is interesting from a scientific point of view. From a practical point of view (regarding sedimentation) only the third type of sediment needs attention since here losses occur and an enhanced understanding of the process (improved sedimentation model) can help to reduce losses and add value. The transition between the three above mentioned sediment types is obviously not sharp and depends as well on the characteristics of the TSHD, but the most interesting range for the particle diameter is roughly between 100 – 300 μm.

Different models were developed in the past to estimate the amount of material lost overboard and the particle size distribution of the sediment in the hopper and in the overflow mixture. These models were based on relative simple models developed for the field of sewage water treatment (Camp, 1946) and were based on an idealised inflow and outflow configuration and a prescribed velocity distribution in the hopper. Because the velocity distribution is prescribed, its relation with the inflow and outflow structures is absent. The second effect is that the concentration distribution does not influence the velocity distribution in these simple models. With the increase in scale of the trailing suction hopper dredgers it is questionable if these simple models were still valid.

The TSHD became the most important type of equipment for the dredging industry in the recent years, but knowledge is still lacking regarding the crucial sedimentation
process onboard this ship. This motivated the VBKO, a branch organisation of the dredging industry (Vereniging van Waterbouwers in Bagger-, Kust- en Oeverwerken) in The Netherlands to initiate a PhD-research programme to increase knowledge on this subject.

The goal of this research programme was to gain more insight into the sedimentation process and to develop a model to simulate this process. The new model should be able to predict the losses to an acceptable accuracy and (although not particularly prescribed) it was expected that many of the restrictions of the existing models could be eliminated.

1.3 Research Methodology and set up of thesis

The research program started in 1997 with a literature survey from which it was concluded that knowledge about the physical process inside the hopper was lacking and the existing models were too simple (Chapter 2).

The second step was the execution of laboratory hopper sedimentation tests with the smallest length scale possible (hence geometry as large as possible to minimise scale effects) at WL|Delft Hydraulics. The model hopper sedimentation tests served three goals:

1. Improved knowledge regarding the sedimentation process in order to formulate a reliable description of the phenomena.
2. Investigation into the influence of the operational parameters (for instance mixture density and discharge), loading and discharge configuration on the sedimentation process.
3. Obtaining high quality data to validate the models to be developed.

To satisfy these goals a lot of effort was spent on measurements (velocity and concentration) inside the hopper beside the usual in- and outflow monitoring (see Section 4.6).

Further to these large geometry laboratory tests fundamental investigations were carried out using a one-dimensional sedimentation column. The influence of sediment concentration, turbulence and the particle size distribution on one-dimensional sedimentation was investigated with these experiments. The influence of air bubbles on the settling velocity of sand particles (at low concentration) was investigated in another sedimentation column. During this test programme the particle size distribution of the grains, and the air concentration was varied. (Chapter 3).

Based on the improved phenomenological description that resulted from the laboratory tests, a one-dimensional vertical (1DV) model was developed. The difference between the earlier models and this new model with its (mostly one-dimensional) predecessors is that the dimension is in vertical direction instead of horizontal (Chapter 6). The 1DV model proved capable of simulating the model hopper sedimentation tests quite well. It
was however unsure whether the model could simulate the process on prototype scale equally well since the horizontal transport in the hopper was not included due to the one-dimensional character.

It was therefore decided to extend the one-dimensional model to two dimensions (two-dimensional-vertical (2DV). At first instance a number of available commercial Computational Fluid Dynamics (CFD) programs were evaluated, but it appeared that none of these programs could be used for the specific hopper sedimentation problem without modification or extension of the (computer) code itself. As a consequence, after a short feasibility study, it was decided to develop a two-dimensional hydrodynamic code specific for the hopper sedimentation process (Chapter 7).

At the same time it was recognised that quantitative information on the influence of the bed shear stress on sedimentation was missing for the conditions present in a hopper (large concentration and velocities below the deposition limit). Consequently special sedimentation tests were carried out in a closed flume. With these test the sedimentation-erosion processes close to the bed could be examined. At first instance pilot tests were carried out at WL|Delft Hydraulics. Later these tests were followed by a test program in the Dredging Technology Research Laboratory of Delft University of Technology with larger dimensions and extensive instrumentation (Chapter 5). From the test results an empirical expression was formulated that is used as a (bed) boundary condition in the 2DV model.

The developed two-dimensional model is compared with benchmark situations in Section 7.12 and with the model hopper sedimentation tests in Section 7.13. To validate the models prototype measurements were executed onboard the TSHD “Cornelia”. These measurements were used to validate both the 1DV and the 2DV model (Chapter 8). Conclusions and recommendations are presented in Chapter 9. In Figure 1.1 a road map for this thesis is shown.
Figure 1.1  Road map for this thesis.
Chapter 2
Sediment movements in a Trailing Suction Hopper Dredger

2.1 Introduction
In this Chapter the trailing suction hopper dredger is introduced in Section 2.2. In Section 2.3 - 2.5 an overview of the different models of the sedimentation process published in literature is given. The Chapter ends with a discussion on the advantages and disadvantages of the different models and the motivation for finding a better description of the process.

2.2 Introduction Hopper Dredger and dredging process
Hopper Dredge
A Trailing Suction Hopper Dredge (TSHD) is a ship that is equipped with one or two suction pipes, which are lowered to the seabed during dredging (Figure 2.2). From the seabed a sand-water mixture is sucked up and discharged into the cargo hold, the so-called hopper. Sand settles in the hopper and the excess water flows overboard. When the hopper is full with sediment, the ship sails to the discharge location where the load is discharged by opening doors in the bottom of the ship or pumped ashore using the dredge pumps of the ship. A cross section of a hopper is shown in Figure 2.4. The bottom section is equipped with sloping walls (like in a silo) to facilitate discharging of the loaded sediment.
Figure 2.2  Trailing Suction Hopper Dredger "HAM 310". The inlet structure varies from ship to ship, but in general the aim is to divide the mixture over the width of the hopper as evenly as possible. For this purpose diffusers or T-shaped spraying pipes are most commonly used. Figure 2.3 shows a top view of a hopper. Two loading pipes are shown together with two overflows.

Figure 2.3  Top view on hopper with loading pipes and overflow structures. When loading starts the hopper is generally partly filled with water, with the water level inside the hopper equal to outside. The mixture is discharged in the hopper using the inlet structures. Sand settles from this mixture and forms a bed, which rises during the loading phase. The excess water is released overboard through the overflow structure, which acts as a weir. These structures are mostly adjustable in vertical position to regulate the overflow level in the hopper. A telescopic overflow structure in the cross section of a hopper is shown in Figure 2.4.
2.2.1 Dimensions of Trailing Suction Hopper Dredgers

The description of a hopper is mostly expressed as the hopper volume (m$^3$) or loading capacity. In Table 2.1 some important properties of an arbitrary selection of the large hopper fleet is shown, ranging from a small hopper (Heron) to a so-called "Jumbo Trailer" (Fairway).

<table>
<thead>
<tr>
<th>Ship</th>
<th>Loading capacity [ton]</th>
<th>Hopper Volume [m$^3$]</th>
<th>Length [m]</th>
<th>Width [m]</th>
<th>Area [m$^2$]</th>
<th>L/W [-]</th>
<th>Discharge [m$^3$/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fairway</td>
<td>31700</td>
<td>23490</td>
<td>87.0</td>
<td>20.4</td>
<td>1784</td>
<td>4.24</td>
<td>14.3</td>
</tr>
<tr>
<td>Amsterdam</td>
<td>23200</td>
<td>18000</td>
<td>66.0</td>
<td>18.0</td>
<td>1188</td>
<td>3.67</td>
<td>13.3</td>
</tr>
<tr>
<td>HAM 310</td>
<td>13652</td>
<td>8225</td>
<td>46.3</td>
<td>15.4</td>
<td>713</td>
<td>3.01</td>
<td>11.3</td>
</tr>
<tr>
<td>Volvox Delta</td>
<td>10390</td>
<td>8143</td>
<td>54.6</td>
<td>14.0</td>
<td>764</td>
<td>3.90</td>
<td>9.4</td>
</tr>
<tr>
<td>Cornelia</td>
<td>7880</td>
<td>6395</td>
<td>52.4</td>
<td>12.6</td>
<td>660</td>
<td>4.16</td>
<td>6.2</td>
</tr>
<tr>
<td>HAM 311</td>
<td>4820</td>
<td>3522</td>
<td>44.0</td>
<td>11.5</td>
<td>506</td>
<td>3.83</td>
<td>4.0</td>
</tr>
<tr>
<td>Heron</td>
<td>1353</td>
<td>974</td>
<td>34.3</td>
<td>7.6</td>
<td>271</td>
<td>4.51</td>
<td>0.8</td>
</tr>
</tbody>
</table>

*Table 2.1 Some Hopper dimensions*
2.3 Existing Models

The sedimentation process in a hopper has a strong resemblance to the sedimentation process in secondary sedimentation tanks used in the field of sewage and water treatment. It is therefore no coincidence that some models used in the dredging industry are based on work done in wastewater treatment. In this respect the work of Camp (1946) must be mentioned since this forms the basis of many subsequent models. Camp defines an ideal settling basin that can be divided into four zones (see Figure 2.5):

1. An inlet zone where the concentration of suspended particles of each size is the same over the vertical at the inlet side of the settling zone.
2. A settling zone where all settling takes place and where the flow is horizontal and the same in all the parts of the settling zone.
3. An outlet zone in which the fluid is collected uniformly over the cross section and directed to an outlet conduit.
4. A sludge zone at the bottom where the particles settle.

Since it is assumed in the model that the horizontal velocity $u$ is everywhere the same throughout the settling zone, all particle trajectories are straight lines in the settling basin. In Figure 2.5, it can be seen that all particles having a settling velocity larger than $v_0$ enters the sludge zone and therefore are removed from the flow. Particles having a smaller settling velocity $w_s$ only enter the sludge zone when they enter the settling zone between point $b$ and $c$. The removal ratio $r_g$ of these particles is therefore equal to the ratio of the lengths $bc$ and $ac$:

$$r_g = \frac{bc}{ac}$$

(Figure 2.5) Ideal Settling Basin according to Camp.
From geometrical considerations follows (with \( L \) as the length of the settling zone and \( H \) as depth of the flow):

\[
\frac{bc}{L} = \frac{w_s}{u} \quad \frac{ac}{L} = \frac{H}{L} = \frac{v_0}{u}
\]  

(2.2)

Combining (2.1) and (2.2), the removal ratio (or grain settling efficiency) for a certain grain size with settling velocity \( w_s \) can be written as:

\[
r_s = \frac{bc}{ac} = \frac{w_s}{v_0}
\]  

(2.3)

Note that for \( w_s > v_0 \) the value of the removal ratio is limited to unity. When \( B \) is the width of the basin and \( Q \) the discharge into the basin, the mean horizontal velocity \( u \) is equal to \( Q/(BH) \). Camp defined the overflow rate \( v_0 \) by substitutions of this expression in (2.2):

\[
v_0 = u \frac{H}{L} = \frac{Q}{BL}
\]  

(2.4)

When a mixture of different particle sizes is supplied to the sedimentation tank the combined removal rate can be computed as follows. Suppose that the cumulative distribution curve of the settling velocities in the mixture is given with Figure 2.6. The value \( p_0 \) in the graph represents a settling velocity equal to the overflow rate (\( w_s = v_0 \)). In that case a fraction of \( (1 - p_0) \) of all particles is removed since the settling velocity of that part is larger than \( v_0 \). The removal rate of a fraction \( dp \) having a settling velocity \( w_s \) is (Figure 2.6):

\[
r_s \ dp = \frac{w_s(p)}{v_0} dp
\]  

(2.5)

Therefore, the total removal of all particles \( r_r \) is:

\[
r_r = 1 - p_0 + \frac{1}{p_0} \int_0^{p_0} w_s(p) dp
\]  

(2.6)

The last term of equation (2.6), the value of the integral is the hatched area in Figure 2.6.
2.3.1 Influence of turbulence

The vertical flux of particles was only based on the settling velocity (advection) due to the assumptions in the previous Chapter. In practical applications the flow in the sedimentation basin is turbulent, hence diffusive transport of particles is also present. The turbulent influence can be included using the advection-diffusion equation for the particles. In this Chapter the contributions of Camp (1946), Dobbins (1944), Groot (1981), and Vlasblom & Miedema (1995) are summarised. The starting point is the two-dimensional advection-diffusion equation for a mono-sized (only one particle diameter present) mixture.

\[
\frac{\partial c}{\partial t} + \frac{\partial (uc)}{\partial x} + \frac{\partial (wc)}{\partial z} = \frac{\partial}{\partial x} \left( \Gamma_{\varepsilon} \frac{\partial c}{\partial x} \right) + \frac{\partial}{\partial z} \left( \Gamma_{\varepsilon} \frac{\partial c}{\partial z} \right)
\]

(2.7)

In which:

\(u\) Horizontal velocity of a particle.

\(w\) Vertical velocity of a particle.

\(\Gamma_{\varepsilon}\) Horizontal diffusion coefficient.

\(\Gamma_{\varepsilon}\) Vertical diffusion coefficient.

The following approximations are introduced:

1. The horizontal velocity of the grains is equal to the horizontal flow velocity.
2. The vertical fluid velocity is zero (ideal settling basin according to Figure 2.5), hence the vertical grain velocity is equal to the settling velocity \(-w_s\).

The transport equation can therefore be simplified to:

\[
\frac{\partial c}{\partial t} + \frac{\partial (uc)}{\partial x} = \frac{\partial}{\partial x} \left( \Gamma_{\varepsilon} \frac{\partial c}{\partial x} \right) + \frac{\partial}{\partial z} \left( \Gamma_{\varepsilon} \frac{\partial c}{\partial z} + w_s c \right)
\]

(2.8)
This equation can be further simplified assuming:
1. Stationary flow \( (\partial c/\partial t = 0) \)
2. Horizontal velocity only a function of vertical position \( (u = u(z)) \)
3. Horizontal diffusion is considered small compared with horizontal advection and therefore neglected.

This leads to the following equation using the continuity equation \( (\partial u/\partial x = 0 \text{ for this situation}) \):
\[
u \frac{\partial c}{\partial x} = \frac{\partial}{\partial z} \left( \Gamma \frac{\partial c}{\partial z} + w_r c \right)
\]
(2.9)

Groot (1981) includes the influence of hindered settling (settling velocity is a function of the concentration \( w_r(c) \)) and writes this equation as:
\[
u \frac{\partial c}{\partial x} = \frac{\partial}{\partial z} \left( \Gamma \frac{\partial c}{\partial z} \right) + w_r \frac{\partial c}{\partial z} + c \frac{\partial w_r}{\partial c} \frac{\partial c}{\partial z}
\]
(2.10)

Groot (1981) assumes a logarithmic distribution for the horizontal velocity. The turbulent diffusion coefficient is based on measurements on model scale (with water, so no inflow of sediment).

2.3.2 Influence of turbulence according to Camp and Dobbins

Camp and Dobbins simplify equation (2.9) further with the following assumptions:
1. The horizontal velocity is constant over depth (uniform distribution).
2. Settling velocity independent of concentration (no hindered settling)
3. Diffusion coefficient constant over depth and horizontal distance

Using these assumptions equation (2.9) becomes:
\[
u \frac{\partial c}{\partial x} = \Gamma \frac{\partial^2 c}{\partial z^2} + w_r \frac{\partial c}{\partial z}
\]
(2.11)

An analytical solution of this partial differential equation was found using separation of variables. In terms of the removal ratio the series type solution is as follows:
\[
r_{rs} = 1 - 8 \left( \frac{w_r H}{2\Gamma} \right)^2 e^x \sum_{n=1}^\infty \frac{\alpha_n^2 b_n e^{i\delta}}{\left( \frac{w_r H}{2\Gamma} \right)^2 + \alpha_n^2 + 2 \frac{w_r H}{2\Gamma} \left( \frac{w_r H}{2\Gamma} \right)^2 + \alpha_n^2}
\]
(2.12)

In which the values of \( \alpha_n \) are the successive real positive roots of the transcendental equation:
\[ 2 \cot \alpha_n = \frac{\alpha_n}{w_s H} - \frac{2\Gamma}{\alpha_n} \frac{w_s H}{2\Gamma} \]

(2.13)

And the exponents \( a \) and \( j_n \) are computed with:

\[ a = \frac{w_s H}{2\Gamma} \quad j_n = -\frac{1}{2a} \left( a^2 + \alpha_n^2 \right) \frac{w_s}{v_0} \]

(2.14)

The coefficient \( b_n = 1 \) when \( \alpha_n \) is in the first or second quadrant and \( b_n = -1 \) when \( \alpha_n \) is in the third or fourth quadrant. Camp plotted the removal ratio as a function of two dimensionless quantities as in Figure 2.7. The lines in the figure are drawn for a constant removal ratio \( r_g \) (without the influence of turbulence). On the horizontal axis the turbulent diffusion is varied. The convergence of the infinite series depends on the value of both \( w_s/v_0 \) and the suspension number \( \frac{1}{2} w_s H / \Gamma \). For values above 10 of the last parameter very slow or no convergence was found, even when hundreds of terms were used in equation (2.12). For these high values the influence of the diffusion is small and equation (2.11) reduces from a diffusion to a wave equation, for which the method used to derive equation (2.12) fails.

![Figure 2.7](image)

**Figure 2.7 Removal ratio including turbulence according to Camp.**

Camp used the following relation between the diffusion coefficient, friction velocity and water depth:

\[ \Gamma = 0.075 Hu_c \]

(2.15)
Using the friction factor \( f \) the relation between the friction velocity \( u_\ast \) and average horizontal velocity \( U \) is written as:

\[
 u_\ast = \sqrt{\frac{f}{8}} U
\]  

(2.16)

Camp arbitrarily uses a friction coefficient \( f \) of 0.024 leading to:

\[
 \frac{w_i H}{2\Gamma} = 122 \frac{w_i}{U}
\]  

(2.17)

2.3.3 Vlasblom and Miedema

Vlasblom & Miedema (1995) have extended the Camp model for the influence of hindered settling, erosion and decreasing flow depth (due to a rising sand bed) with time. The last effect is a specific situation for the hopper sedimentation application. For the situation where the Camp model was originally developed the flow depth in the settling zone remained constant since the settled sludge was mechanically removed. The total removal ratio for a certain grain was written by Vlasblom & Miedema as the product of the grain settling efficiency and a correction factor that takes the influence of turbulence into account:

\[
 r_c = \frac{1}{r} \int_0^1 r_s dp
\]  

(2.18)

From curve fitting, an expression for factor \( r_c \) was derived in such a way that the product reproduced the computed \( r_s \), values of Camp (Figure 2.7). Using equation (2.17) the quantity \( w_i/U \) is used for the influence of the horizontal velocity. The expressions of Miedema & Vlasblom (1996) are given here since these match more closely the results of equation (2.11). For \( r_s = w_i/v_0 < 1 \) the factor \( r_c \) was computed with:

\[
 r_c = 1 - 0.184r_s^{0.88-0.2r_s} \left[ 1 - \tanh \left( r_s^{-0.13-0.8r_s} \left( \log \left( \frac{w_i}{U} \right) + 0.5 + r_s^{-0.33-0.94r_s} \right) \right) \right]
\]  

(2.19)

For values of \( r_s > 1 \) the following expression was proposed:

\[
 r_c = r_s^{-1} \left[ 1 - 0.184r_s^{-0.69-0.38r_s} \left[ 1 - \tanh \left( r_s^{0.77-0.8r_s} \left( \log \left( \frac{w_i}{U} \right) + 0.5 + r_s^{1.01-0.18r_s} \right) \right) \right] \right]
\]  

(2.20)

Note that in Miedema & Vlasblom (1996) the last term in equation (2.19) and (2.20) has the wrong sign. The resulting figures in this publication are nevertheless correct. It is clear that these formulas are much easier to implement than equation (2.12) and the convergence problems are avoided as well. The influence of a different friction factor can be taken into account by the following formulation:
Influence of erosion

The flow depth in a hopper decreases over time due to the rising sand bed. As a result the horizontal flow velocity increases with time and hence a lower value for the suspension number is computed using equation (2.21). Figure 2.7 shows that this leads to a lower value of the removal ratio. Owing to the larger horizontal flow velocity the shear stress acting on the bed increases as well, implying a reduction of sedimentation. Further to Camp (1946), Vlasblom and Miedema (1995) implement the influence of erosion by defining a “scour velocity” $u_s$:

$$u_s = \sqrt{\frac{8(1-n_v)\mu d BD}{f}}$$

In which:

- $f$ Friction factor
- $\Delta$ Relative sediment density $(p_r - p_w)/p_w$
- $n_v$ Porosity of the sand bed
- $g$ Gravitational acceleration
- $D$ Grain diameter

This velocity is not to be mistaken with an erosion velocity. It is a threshold flow velocity at which grains with diameter $D$ do not settle in the bed anymore. This threshold velocity can be compared with a critical Shields value. The coefficient $\mu$ depends on the internal friction of the sediment and has a value in the range of 0.1-0.6 for sand grains (according to Vlasblom & Miedema, 1996).

The influence on the removal rate is computed as follows. Suppose that on a certain moment the flow velocity in the basin equals $U$. Application of equation (2.22) using $u_s = U$ leads to a threshold grain diameter $D_{cr}$.

$$D_{cr} = \frac{f}{8(1-n_v)\mu d g U^2}$$

All grains smaller than this size do not settle in the hopper (according to the Vlasblom & Miedema model). A critical settling velocity $w_{cr}$ can be determined using this critical diameter. When $p_r$ is the value in the cumulative settling curve corresponding to this critical settling velocity (see Figure 2.6), the total settling efficiency including erosion becomes:

$$r_e = \int_{p_r}^{1} r_r r_g dp$$

(2.24)
An alternative approach to take account of the influence of erosion is analysing the sum of the settling and pickup rate. This method is shown in the next section.

2.3.4 Implementation of erosion according to Van Rhee

Van Rhee (2001) implemented an alternative method to include the influence of erosion in the Vlasblom and Miedema model. Instead of using a threshold velocity Van Rhee computed the settling flux and decreased this value with a pickup flux using the pickup function of Van Rijn (1984). According to Van Rijn the pickup rate \( E \) is a function of grain size and bed shear stress as follows:

\[
\frac{E}{\rho_s \sqrt{\Delta g D}} = 0.00033D_0^{3.3}T^{1.5}
\]

(2.25)

In which \( D_0 \) is a dimensionless grain diameter defined as (with \( \nu \) as kinematic viscosity):

\[
D_0 = \frac{\sqrt{\Delta g}}{\nu^2}
\]

(2.26)

\( T \) is a transport parameter depending on the bed shear stress. Van Rijn used the following expression:

\[
T = \frac{\theta - \theta_c}{\theta_c} ; \quad \theta = \frac{u^2}{\Delta g D}
\]

(2.27)

Where \( \theta \) is the Shields parameter and \( \theta_c \) the critical Shields parameter.

The influence of the alternative formulation for erosion on the original Vlasblom & Miedema model was however small. This is caused by a very important assumption in all models described so far: The logarithmic or uniform velocity distribution. The mean horizontal flow velocity equals:

\[
U = \frac{Q}{BH}
\]

(2.28)

Using the definition of the overflow rate (equation (2.4)):

\[
\frac{U}{v_0} = \frac{L}{H}
\]

(2.29)

The overflow rate is of the same order as the settling velocity of the grains. For a hopper the length / depth ratio is less than ten at the start of the loading phase (no sediment in the hopper). During loading the sediment level rises, hence the flow depth decreases, but during a long time the (assumed) horizontal velocity is relative low, about ten times the settling velocity of the grains. This is well below threshold "scour velocity" or below the critical Shields parameter. This means that no re-suspension takes place
according to these models, apart from the final loading stage where the height of the settling zone is very small.

2.4 Modelling of the decreasing flow depth in the settling zone

In the models for the sewage treatment plants the height of the settling zone is more or less constant in time because the settled sludge is constantly removed. The situation for a hopper dredge is considerably different. The objective is to fill the settling zone completely within a relative short time. In the Vlasblom & Miedema (1995, 1996) model the following procedure was followed. The balance of the in- and outflowing solids reads:

\[
\frac{dM_{so}}{dt} = \rho_s (Q_{in}c_{in} - Q_o c_o) = \rho_s c_{in} \left( Q_{in} - (1 - r) Q_o \right) \tag{2.30}
\]

In which:

- \(M_{so}\) Mass of solids in the hopper.
- \(Q_{in}, Q_o\) In- and outflowing discharge.
- \(c_{in}, c_o\) In- and outflowing (volume) concentration.

Vlasblom & Miedema calculated \(r\) with equation (2.24). Van Rhee (2001) used the pickup function of Van Rijn (equation (2.25)). The solids mass removed by erosion in the hopper is:

\[
\frac{dM_{so,e}}{dt} = EBL \tag{2.31}
\]

Using this relation the mass balance can be written as:

\[
\frac{dM_{so}}{dt} = \rho_s \left( Q_{in}c_{in} - Q_o c_o \right) = \rho_s c_{in} \left( Q_{in} - (1 - r) Q_o - \frac{E \cdot BL}{\rho_s c_{in}} \right) \tag{2.32}
\]

The removal ratio is in this case computed with equation (2.6), since the influence of erosion is taken into account with the last term of (2.32). In the model it is assumed that the total mass of sand in the hopper is stored in the bed, so the possible storage in suspension is neglected. The average level of the bed \(H_b\) can therefore directly be related to \(M_{so}\):

\[
H_b = \frac{M_{so}}{\rho_s (1 - n_b) BL} \tag{2.33}
\]

The depth \(H\) is the difference between the original (empty hopper) height \(H_0\) and the bed level \(H_b\). The loading process can now be simulated by numerical integration of equation (2.32). At every time step the height \(H\) can be computed and likewise \(U\), \(r\), and \(E\). At the end of the loading process the height of the settling zone becomes small.
and hence the horizontal flow velocity $U$ is large. As a result the term between the large brackets in equation (2.32) decreases to zero. The outflux of sand is equal to the influx.

### 2.4.1 Dynamic extension of Ooiijens

In the Vlasblom and Miedema model the concentration in the hopper is always equal to the inflow concentration and the outflow concentration responds instantaneously on the calculated settling efficiency. Ooiijens (1999) adds the time effect by regarding the hopper as an ideal mixing vessel. The calculated concentration in the hopper is used for the settling efficiency calculation. The extension is an evident improvement, since it adds for instance the influence of overflow level variation on the calculation. The basis of the method is however still the Camp theory i.e. based on a strongly simplified flowfield.

### 2.5 Yagi’s Model

The first hopper sedimentation model developed especially for the dredging industry was published by Yagi (1970) on the first WODCOM Conference. Unlike models previously discussed that were based on work in the sanitary industry, this model was based on the concentration distribution in open channel flow. Yagi began with a definition of the average loading efficiency as:

$$\eta_m = \frac{G_H}{G_P} = \frac{B \int_0^t g_u dt}{B \int_0^t g_o dt}$$  \hspace{1cm} (2.34)

Where $G_H$ is the total quantity of solids in the hopper (settled and in suspension) and $G_P$ is the total amount of solids pumped into the hopper and $g_u$ and $g_o$ are the same quantities per unit time and width of the hopper. Another important parameter is the cumulative overflow loss, which is the ratio between the total volume of sediment lost overboard and the total amount of sediment pumped in the hopper. The cumulative overflow loss is defined as:

$$OV_{\text{cum}}(t) = \frac{\int_0^t c_o(t) Q_o(t) dt}{\int_0^t c_o(t) Q_m(t) dt}$$  \hspace{1cm} (2.35)

Where $c_o$ equals the inflow concentration, $c_o$ the overflow concentration, $Q_m$ the discharge in the hopper and $Q_o$ the overflow discharge. The relation between the average loading efficiency and the cumulative overflow loss is found by combining equations (2.34) and (2.35).
\[ \eta_m = 1 - OV_{cum} \]  
(2.36)

If the discharge of sediment into the hopper is assumed to be constant, equation (2.34) can be rewritten as:

\[ \eta_m = \left( \frac{\int \frac{g_{Ht}}{t} \, dt}{\int \frac{g_p}{t} \, dt} \right) \frac{\int \eta dt}{t} \]  
(2.37)

Yagi assumes that the sediment distribution in a hopper can be compared with the (uniform) situation in open channel flow. The concentration in that case can be described using the one-dimensional advection-diffusion equation:

\[ \Gamma \frac{dc}{dz} + w_c c = 0 \]  
(2.38)

This expression follows from equation (2.9) when the sediment distribution is uniform. For the value of the diffusion coefficient \( \Gamma \), Yagi assumed a parabolic distribution. The depth averaged value reads:

\[ \bar{\Gamma} = \frac{1}{2} \kappa u, H = \beta \kappa u, H \]  
(2.39)

In which \( \kappa \) is the Von Kármán constant and \( H \) the flow depth. Using the constant value for the diffusion coefficient an analytical solution can be obtained easily and expressed as a function of the dimensionless suspension number \( Z \) and dimensionless height \( \xi \).

\[ \frac{c}{c_b} = e^{-Z \xi} \]

\[ Z = \frac{w_c}{\beta \kappa u} = \frac{w_c H}{\bar{\Gamma}} \]  
(2.40)

\[ \xi = \frac{z}{H} \]

An important parameter is the unknown concentration near the settled sediment bed \( c_b \) (near bed concentration). A relation between the near bed concentration and inflow concentration is expected. Yagi assumes that the depth-averaged concentration in the hopper (in the suspension above the bed) is equal to the inflow concentration \( c_{in} \).

\[ c_m = \int_0^1 c_b e^{-Z \xi} d\xi \quad \Rightarrow \quad c_b = \frac{c_{in} Z}{1 - e^{-Z}} \]  
(2.41)

Substitution of this result in equation (2.40) leads to the following relationship between the inflow concentration and the concentration distribution in the hopper:

\[ \frac{c}{c_{in}} = \frac{Z}{1 - e^{-Z}} e^{-Z \xi} \]  
(2.42)
The relationship between this distribution and the hopper efficiency is the next step. Equation (2.36) shows the relation between the cumulative overflow losses and the (time) averaged hopper efficiency. A similar relationship exists for the overflow flux and the hopper efficiency $\eta$ (efficiency at a certain point in time, not time averaged). Note that the loading efficiency is equivalent to the removal ratio defined in the previous Section.

$$\eta = 1 - \frac{Q_{c_{in}}}{Q_{m}c_{in}}$$  \hspace{1cm} (2.43)

Yagi computes the amount of sediment $g_o$ per unit width in the overflow by integration of the concentration profile between the overflow- and water level:

$$g_o = \int_{-h-h}^{h} c(z)u_{i}dz = \int_{-\xi_{h}}^{1} c_{in}u_{i}H \frac{Z}{1-e^{-Z}} e^{-Z\xi}d\xi$$  \hspace{1cm} (2.44)

Where

$h$ Difference between water level and overflow level.

$\xi_h$ Dimensionless value of $h$: $\xi_h = h / H$.

$u_{i}$ Flow velocity in overflow.

Integration yields:

$$g_o = c_{in}u_{i}H \frac{e^{-Z(1-\xi_h)} - e^{-Z}}{1-e^{-Z}}$$  \hspace{1cm} (2.45)

Yagi now states that the term $c_{in}u_{i}H$ is equal to the influx of sand $g_p$ and writes the outflux as:

$$g_o = \frac{g_p}{1-e^{-Z}} \{e^{-Z(1-\xi_h)} - e^{-Z}\}$$  \hspace{1cm} (2.46)

Substitution of this result in the definition of the loading efficiency (equation (2.43)) yields:
\[ \eta = \frac{1 - e^{-\frac{Z(1 - \xi_h)}{1 - \xi_h}}}{1 - e^{-Z}} \]  

(2.47)

Finally the term \( Z(1 - \xi_h) \) is rewritten by substitution of \( Z \) and the following definition of the friction velocity (using a different friction coefficient as usual):

\[ u_* = \sqrt{\frac{f}{2}} \cdot U_H \]  

(2.48)

Where \( U_H \) is the mean horizontal flow velocity. The following expression is obtained:

\[ \eta = \frac{1 - e^{-\alpha \frac{u_*}{U_H}}}{1 - e^{-Z}} \]  

\[ \alpha = \delta \frac{\sqrt{f}}{\beta} \sqrt{\frac{1}{2}} (1 - \xi_h) \]  

(2.49)

Yagi defines \( \delta \) as a "lifting soil coefficient which shows the degree of soils lifted by the increase of \( U_H \) when \( H \) decreases". Thus \( \delta \) is an expression for the erosion effect under flow. Using experimental results Yagi concludes that \( \alpha = 2.5 \), hence constant although equation (2.49) shows a dependency of the water level on the overflow. The explanation of Yagi of this constant value is that the reduction of \((1 - \xi_h)\) during loading is counterbalanced by an increasing value of \( \delta \). The expression of the loading efficiency can be further simplified for sand where the term \( e^{-Z} \) is small compared with unity:

\[ \eta = 1 - e^{-\alpha \frac{u_*}{U_H}} \]  

(2.50)

### 2.5.1 Discussion

Yagi computes the outflow by integration of the velocity and concentration profile between overflow- and water level and eliminates the term \( c_w u_h H \) with the influx of sand \( g_0 \). This implies that it is assumed that the velocity in the overflow is equal to the mean velocity in the hopper which is not correct. Using continuity the value of the velocity in the overflow is approximately:

\[ u_h = \frac{H}{h} U_H \]  

(2.51)

Which leads to a more complicated relation for the hopper efficiency:

\[ \eta = \frac{1 - \frac{H}{h} e^{-Z(1 - \xi_h)} - \left(1 - \frac{H}{h}\right) e^{-Z}}{1 - e^{-Z}} \]  

(2.52)
2.5.2 Average loading efficiency

It is important to note that the outcome of Yagi’s model represents the hopper efficiency (or removal ratio in Camp’s model) at a certain time only. Due to the dependence of (2.50) of \( U_H \), the removal ratio decreases during time. Thus the average hopper efficiency does not follow directly from equation (2.50). Deriving an expression for the average hopper efficiency can however complete the original work of Yagi. This can be accomplished using the following relation between the mixture level in the hopper and the efficiency:

\[
\frac{dH}{dt} = -\eta \frac{g_p}{(1-n_0)L}
\]  
(2.53)

Substitution of (2.50) in this expression using \( U_H = Q/(BH) \) yields:

\[
\frac{dH}{dt} = -\left(1-e^{-\alpha \omega_B BH / Q}\right) \frac{g_0}{(1-n_0)L}
\]  
(2.54)

Integration gives:

\[
\int_{H_0}^{H} \frac{dH}{1-e^{-\alpha \omega_B BH / Q}} = -\int_{0}^{t} \frac{Qc_{in}}{(1-n_0)BL} \, dt
\]  
(2.55)

This leads to the following expression for the loading time \( t \) needed to fill the hopper to a sand level of height \( H_{in} \):

\[
t = \frac{(1-n_0) L}{c_{in} \alpha \omega_s} \ln \left( \frac{-1+e^{-\alpha \omega_B BH / Q}}{-1+e^{-\alpha \omega_s B / Q} e^{-\alpha (H_{in}-H)}} \right)
\]  
(2.56)

\[
a = \alpha \omega_s B / Q
\]

When the minimum time (hence with overflow losses equal to zero) needed to fill the hopper with a height \( H_{in} \) is defined as \( t_{min} \), the relation between the actual loading time and the average hopper efficiency and cumulative overflow loss follows from:

\[
\frac{t}{t_{min}} = \frac{1}{1-OV_{cam}} = \frac{1}{\eta_m}
\]  
(2.57)

The expression for the minimum time follows directly from the sediment volume balance:

\[
t_{min} Qc_{in} = (H_0-H)(1-n_0)BL \Rightarrow t_{min} = \frac{(H_0-H)(1-n_0)BL}{Qc_{in}}
\]  
(2.58)

Combining equations (2.56), (2.57) and (2.58) the following expression for the average loading efficiency can be found:
\[ \eta_m = -a (H_0 - H) \frac{1}{\ln \left( \frac{-1 + e^{-aH}}{-1 + e^{-aH} e^{-a(H_0 - H)}} \right)} \] (2.59)

The average loading efficiency (or cumulative overflow loss) is a function of \( a, H_0 \) and \( H \) and does not depend on the length of the hopper \( L \). This is not realistic and is a direct result of the simplifications in the model.

2.6 Shortcomings of the different simple models for the application in Dredging Industry

Most of the models described above provide reasonable values for the overflow losses and hopper efficiency, but are limited because:

- All hopper sedimentation models are based on a simplified velocity distribution in the hopper where the inflow and outflow section are simplified or idealised. This implies that the influence of the inflow and overflow constructions (position, level, quantity) on the sedimentation is not accounted for and cannot be simulated.
- In the models the velocity distribution is assumed to be uniform or logarithmic. It is questionable if this assumption is correct when these models are applied for hopper sedimentation where high mixture densities are discharged in the hopper. This leads most probably to density currents. In that case the flow is concentrated near the bed. Velocities in the density current are much higher compared with the uniform or logarithmic distribution. This affects the generation of turbulence and erosion. The possibility of the presence of density currents in the hopper was already mentioned by De Koning, 1977.
- The influence of the flow velocity or bed shear stress on sedimentation is modelled in a very simple way.
- Some models do not include the influence of hindered settling and when they do the interaction of the different fractions is too simple for high concentrations.
- Most models do not include the influence of the variation of the bed level or the variation of the water level during the filling process.

2.7 Hydrodynamic Models

The need to prescribed the velocity distribution and to idealise the inflow and outflow configuration was recognised as a shortcoming in the field of water treatment as well. The rapid increase of personal computer power and developments in the field of CFD (Computational Fluid Dynamics) in late 80's and 90's enabled more sophisticated approaches to determine the flow pattern and settling process in a sedimentation basin. Two-dimensional numerical models were developed. Reference is made to Schamber & Larock, 1983, Devantier & Larock, (1986), Rodi & Adams (1989) and Zhou &
McCorquodale, (1992, 1993), Mazzolani et al. (1998). The models were based on the 2D Reynolds Averaged Navier-Stokes equations with a $k-\varepsilon$ turbulence model. The sediment transport was modelled using the advection-diffusion transport equation. Rodi & Adams included the influence of the particle size distribution by dividing the complete spectrum of grain sizes in $n$ groups of constant particle size. The total concentration in the settling basin is rather small, so the influence of hindered settling or interaction between the different groups can be neglected. Concentration variation leads to differences in density and hence buoyancy effects in the fluid. Coupling of the sediment transport equations with the momentum equation was commonly accomplished using the Boussinesq approach (see par. 7.3.2). In settling tanks of clarifiers the sediment is mostly very fine and flocculation and consolidation is of importance unlike the situation in a hopper dredger. The vertical position of the water surface and bed does not vary in the settling tanks, so no special arrangements have to be included in the models to deal with moving boundaries of the computational domain. These hydrodynamic models are an evident improvement over the simple models since the flow pattern, and accompanying shear stresses and turbulent diffusion is much more in accordance with reality. The models cannot be applied to the sedimentation process in a hopper dredger however, because some important features of that specific sedimentation process are still not included:

- Variation of the water level due to filling of the hopper in the first loading phase and/or by variation of the overflow level.
- Variation of the sediment bed level.
- Large Concentration.
- Grading of the sediment.
- Relative short filling time of the hopper, hence the flow field does not reach a stationary situation.
- Influence of the bed shear stress on sedimentation velocity valid for the soil types relevant for the dredging industry.

In this Chapter the hydrodynamic models were not dealt with in detail since the section on the two-dimensional hopper sedimentation model is completely devoted to that subject.
Chapter 3

Settling velocity of sediments

3.1 Introduction

In all hopper sedimentation models the settling velocity of the grains is an important parameter. The one-dimensional settling of grains is therefore analysed in detail in this Chapter. To start with, the settling velocity of a single grain is considered. Thereafter the influence of concentration, particle size distribution, clay fraction and the influence of air bubbles on the settling velocity is analysed. The Chapter is closed with the definition of the sedimentation velocity.

3.2 Settling velocity of a single grain

When a single grain settles in a stagnant fluid and interactions other than inertia and drag are neglected, the equation of motion is given by:

\[ V_s \frac{dw}{dt} (\rho_s + C_m \rho_w) = V_s (\rho_s - \rho_w) g - C_D A_g \frac{1}{2} \rho_w w^2 |w| \]  

(3.1)

Where:
- \( V_s \) Volume of the grain.
- \( w \) Vertical velocity of the grain.
- \( A_g \) Projected surface of the grain.
- \( C_D \) Drag coefficient.
- \( C_m \) Added mass coefficient.

The volume of the grain can be written as:

\[ V_s = \psi \frac{\pi}{6} D^3 \]  

(3.2)

Where \( \psi \) is a shape factor that takes the deviation from a perfect sphere (with diameter \( D \)) into account. The area subject to flow resistance is equal to \( A_g = \pi \sqrt{D^2} \). The drag coefficient \( C_D \) depends on the particle Reynolds number \( Re_p \), defined with:

\[ Re_p = \frac{wD}{\nu} \]  

(3.3)

Where \( \nu \) is the kinematic viscosity of the fluid. When the initial vertical velocity is equal to zero, the grain accelerates until the buoyancy is in equilibrium with the drag resistance and the terminal velocity \( w_0 \) is reached. The expression for the terminal (or
equilibrium) velocity follows directly from the equation of motion (acceleration equal to zero):

\[ w_0 = \frac{\sqrt{4gD\psi}}{3C_D} \]  

(3.4)

With \( \Delta \) as the specific density defined as \( \left( \rho_s - \rho_w \right) / \rho_w \). Several empirical relationships between the drag coefficient and the Reynolds number for a sphere are in use. The most common formulae for the laminar (or Stokes), transitional and turbulent regime are respectively:

\[ C_D = \frac{24}{Re_p} \quad Re_p \leq 1 \]

\[ C_D = \frac{24}{Re_p} + \frac{3}{\sqrt{Re_p}} + 0.34 \quad 1 < Re_p < 2000 \]  

(3.5)

\[ C_D = 0.4 \quad Re_p \geq 2000 \]

Substitution of the above formula for the drag coefficient for the laminar and turbulent regime leads to an explicit relation for the settling velocity. For the laminar regime (\( Re_p \leq 1 \)) the result is called the Stokes settling velocity:

\[ w_0 = \frac{\sqrt{\Delta gD^2}}{18\nu} \]  

(3.6)

For the turbulent regime (\( Re_p \geq 2000 \)):

\[ w_0 = 1.8\sqrt{\Delta gD\psi} \]  

(3.7)

For the transition regime the settling velocity must be computed by iteration of the drag coefficient in case equation (3.5) is used. Alternatively empirical relationships can be used as well for this regime. The equation of Budryck is mentioned, valid for 0.1 mm < D < 1 mm:

\[ w_0 = 8.925\sqrt{1 + 95 \left( \rho_s - \rho_w \right) \frac{D^3}{100\nu^2} - 1} \]  

(3.8)

The constants in the equation are not dimensionless. The following dimensions must be used for the variables in this strange equation: \( w_0 \) in mm/s, D in mm and the densities in ton/m\(^3\). Another formula valid for the transitional regime (Ruby & Zanke):

\[ w_0 = \frac{10\nu}{D} \left[ \sqrt{1 + \frac{\Delta gD^3}{100\nu^2} - 1} \right] \]  

(3.9)

The different methods for the transition regime are compared in Figure 3.1. For the shape factor \( \psi \), a value of 0.7 was chosen. The kinematic viscosity of water depends on
the temperature. The following relation between the kinematic viscosity in [m²/s] and temperature $T$ in degrees Celsius is often used:

$$
\nu = \frac{40 \cdot 10^{-6}}{20 + T}
$$

(3.10)

The value for the viscosity used for Figure 3.1 is $1.11 \cdot 10^{-6}$ m²/s.

![Diagram](image)

**Figure 3.1** Settling velocity versus grain diameter for the laminar-turbulent transition.

It is clear that the empirical relations differ from the theoretical value based on iteration of the drag coefficient.

### 3.3 Particle Response Time – Stokes number

When equation (3.1) is solved the time needed to reach the terminal velocity can be determined. Since this value is reached asymptotically, it is practical to compute the time needed to obtain, say 99% of this value. In Figure 3.2 the settling velocity and distance versus time is plotted for different grain diameters. It is clear that for these fine sands the terminal velocity $w_0$ is reached within a very short distance (order of magnitude is one grain diameter) and time (end of the line in left panel).
An important dimensionless parameter to relate the grain response time $\tau_g$ to the flow field is the Stokes number, defined as the ratio between the grain response time and a hydrodynamic time scale $\tau_h$.

$$St = \frac{\tau_g}{\tau_h} = \frac{\tau_g \mu}{L}$$

(3.11)

The hydrodynamic time scale is as usual composed of the ratio of a velocity and length scale. The Stokes number is important to characterise the behaviour of grains in a fluid. For very large Stokes numbers $St \gg 1$ the grains are not influenced by the flow fluctuations and the behaviour is dominated by bouncing against the walls and interparticle collisions. For small Stokes numbers $St \ll 1$ a particle reacts almost instantaneously to velocity fluctuations and the particle follows the flow. In Chapter 7 it is shown that the Stokes number determines the computational approach for multiphase flows.

### 3.4 Influence of the concentration

So far only the behaviour of a single grain in a fluid is analysed. When grains settle in a suspension the mutual influence of the grains decreases the settling velocity. This effect is called hindered settling. The hindered settling effect is caused by the returning water flow, the increased pressure gradient in the water due to the increased mixture density of the fluid and inter-particle collisions.

The influence of the concentration is generally written as:

$$w_s = f(c)w_0$$

(3.12)

Where $c$ is the volume concentration of the sediment. The settling velocity of the grains in a suspension is the product of the settling velocity of a single grain and a hindered settling function. The relation of Richardson & Zaki (1954) is well known. The following empirical function was proposed based on experiments:
\begin{equation}
  f(c) = (1-c)^n
\end{equation}

The exponent is a function of the particle Reynolds number \( \text{Re}_p \). Based on experiments with concentrations \( 0.05 < c < 0.65 \) and Reynolds numbers \( 0.000185 < \text{Re}_p < 7150 \), Richardson & Zaki found:

\begin{align*}
  \text{Re}_p < 0.2 & \quad n = 4.65 \\
  0.2 \leq \text{Re}_p < 1 & \quad n = 4.35 \text{Re}_p^{0.03} \\
  1 \leq \text{Re}_p < 200 & \quad n = 4.45 \text{Re}_p^{0.1} \\
  \text{Re}_p \geq 200 & \quad n = 2.39
\end{align*}

(3.14)

For low Reynolds numbers (laminar) or high Reynolds numbers the exponent of the hindered settling function is constant. In between a transition zone exists where the exponent varies. The description of the exponent with four different functions does not lead to a smooth behaviour. A smooth presentation can be achieved using a logistic curve:

\begin{equation}
  \frac{A-n}{n-B} = C\text{Re}_p^\alpha
\end{equation}

(3.15)

Or, more conveniently:

\begin{equation}
  n = \frac{a + b\text{Re}_p^\alpha}{1+c\text{Re}_p^\alpha}
\end{equation}

(3.16)

For the coefficients a, b, c and \( \alpha \) the following values are reported (Table 3.1):

<table>
<thead>
<tr>
<th>Author(s)</th>
<th>( \text{Re}_p )</th>
<th>Conc.</th>
<th>a</th>
<th>b</th>
<th>c</th>
<th>( \alpha )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Garside (1977)</td>
<td>( 0.001 \leq \text{Re}_p &lt; 3 \cdot 10^4 )</td>
<td>( 0.04 &lt; c &lt; 0.55 )</td>
<td>5.1</td>
<td>0.27</td>
<td>0.1</td>
<td>0.9</td>
</tr>
<tr>
<td>Rowe (1987)</td>
<td>( 0.2 &lt; \text{Re}_p &lt; 10^3 )</td>
<td>( 0.04 &lt; c &lt; 0.55 )</td>
<td>4.7</td>
<td>0.41</td>
<td>0.175</td>
<td>0.75</td>
</tr>
<tr>
<td>Di Felice (1999)</td>
<td>( 0.01 &lt; \text{Re}_p &lt; 1 \cdot 10^3 )</td>
<td>( 0 &lt; c &lt; 0.05 )</td>
<td>6.5</td>
<td>0.3</td>
<td>0.1</td>
<td>0.74</td>
</tr>
</tbody>
</table>

\textit{Table 3.1} Overview coefficients for hindered settling function.

The different approaches are compared in Figure 3.3. It is clear that the expression according to Rowe (1987) is a smoothed representation of the original Richardson and Zaki relations. Using the coefficients of Garside & Al-Dibouni (1977) the same trend is shown at a somewhat higher value for the exponent.
Using the values according to Di Felice (1999) very high values for the exponent are found. This relation is however only valid for dilute mixtures (c < 5%).

3.5 The influence of the particle size distribution

The formulae above predict settling velocities for mono disperse suspensions (only one grain diameter present). For a poly-disperse suspension another approach must be followed. The simplest approach is to use the total concentration in the reduction function to calculate the settling velocity for a certain fraction. When the PSD is approximated using $N$ different particle sizes and the concentration of a certain size is $c_i$, the settling velocity of that fraction reads:

$$w_{s,i} = w_{0,i} (1-c_i)^n, \quad c_i = \sum_{i=1}^{N} c_i$$

(3.17)

The exponent $n_i$ corresponds with the particle Reynolds number of the subject fraction. This approach is however not correct because the effect of the return flow of large particles on the small particles is not included. So when this simple relation is used all particles move in the same direction, while in reality it is possible that small particles move in opposite direction due to the return flow of the large particles. When the effect of the grain size is to be modelled correctly a more sophisticated approach is needed. A better approach is to assume that every grain settles with a certain slip velocity relative to the fluid velocity $v_w$:

$$w_{s,i} = v_w - v_{s,i}$$

(3.18)
According to Mirza & Richardson (1979) the slip velocity of a fraction $i$ can be calculated with:

$$v_{s,i} = w_{0,i} (1 - c_i)^{n_i - 1}$$  \hspace{1cm} (3.19)

This follows directly from the hindered settling equation since the settling particles create a return flow that has to flow through an area $1 - c_i$.

In this approach the influence between two or more different fractions is accounted for through the total concentration and the return flow of all particles. The particle-particle interactions between different fractions are however not included and as a result this approach does not give good agreement with experimental data for particles with large differences in size (or density). A relative simple method to include the interparticle influences for different fractions was proposed by Selim, et al. (1983). In the settling velocity $w_0$ the effect of the presence of the smaller grains on the settling velocity of the larger grains is taken into account by a correction of the specific density for these grains. According to Selim, et al. (1983)) the settling velocity of a fraction with grain diameter $d_k$ follows from:

$$w_{0,k} = f(\Delta_k, d_k, ......)$$  \hspace{1cm} (3.20)

The specific density for grain size $k$:

$$\Delta_k = \frac{\rho_i - \rho_{\text{sp},k}}{\rho_{\text{sp},k}} \hspace{1cm} \rho_{\text{sp},k} = \rho_i \sum_{i=1}^{n_k} c_i + \rho_{\omega} \left( 1 - \sum_{i=1}^{n_k} c_i \right)$$  \hspace{1cm} (3.21)

In other words: It is assumed that a grain with a certain size settles in a suspension formed by the grains with a smaller size. A drawback of this specific density correction is that it can produce inaccurate results when the particle size distribution is close to uniform as is shown in Appendix D.

A theory for predicting sedimentation velocities in polydisperse dilute suspensions of spheres at low Reynolds number was developed by Bachelor, 1982. The reduction factor due to hindered settling for a certain fraction is computed using:

$$f_i(c_i) = 1 + \sum_{j=1}^{N} S_{ij}c_j$$  \hspace{1cm} (3.22)

The (negative) dimensionless sedimentation coefficients $S_{ij}$ are functions of the radius ratio of the fractions $\lambda_{ij} = D_j/D_i$ the buoyant density ratio $\gamma = (\rho_j - \rho)/(\rho_i - \rho)$ and the Péclet number defined as:

$$Pe = \frac{Dw_0}{\varepsilon_0}$$  \hspace{1cm} (3.23)

In which $\varepsilon_0$ is the Brownian diffusivity. In the case of sedimentation of sand this type of diffusion does not play any role ($Pe \gg 1$). In the common case that all particles have
the same density and (\( \gamma = 1 \)) and large Péclet number the values of \( S_{ij} \) can be calculated with the following expression (Davis & Gecol, 1994):

\[
S_{ij} = -3.50 - 1.10 \lambda_{ij} - 1.02 \lambda_{ij}^2 - 0.002 \lambda_{ij}^3
\]  

(3.24)

For dilute suspensions Bachelor’s theory is in good agreement with experimental data. Unfortunately, the theory does not hold good for concentrated suspensions. A new hindered settling function without empirical parameters was proposed by Davis & Gecol in 1994 for polydisperse suspensions with arbitrary sizes, densities, and concentrations at low Reynolds number:

\[
f_i(c_i) = (1-c_i)^{S_{ii}} \left[ 1 + \sum_{j \neq i} (S_{ij} - S_{ii}) c_j \right]
\]  

(3.25)

In equation (3.25) the last term is summed for \( j = 1 \) to \( N \), except for \( i = j \) (\( N \) is number of fractions). It will be shown that the key features of this expression are that it agrees with the Bachelor theory in the dilute limit and that it reduces to the Richardson & Zaki correlation for monodisperse suspensions with \( n \) replaced by \(-S_{ii}\) (Davis & Gecol, 1994). The reduction to the Richardson & Zaki expression for a monodisperse suspension is caused by the zero value of the term between the large brackets reducing Equation (3.25) to:

\[
f(c) = (1-c)^{5.622}
\]  

(3.26)

Using (3.24), Equation (3.25) can be written as:

\[
f_i(c_i) = (1-c_i)^{5.622} \left[ 1 + \sum_j S_{ij} c_j - 5.622 \sum_j c_j \right]
\]  

(3.27)

Or:

\[
f_i(c_i) = (1-c_i)^{5.622} \left( 1 + \sum_j S_{ij} c_j \right) - 5.622 c_i (1-c_i)^{5.622}
\]  

(3.28)

The last term of (3.28) vanishes and the term \((1-c_i)^{5.622}\) approaches unity for dilute mixtures \((c_i << 1)\), hence the expression of Bachelor (3.22) is obtained for that case. Several formulae have been summarised above for a certain grain size. To compute the settling velocities for all grain sizes the combined action of all grain sizes must be quantified. This can be done using the volume balance in vertical direction for both sand and water.

\[
\sum_{i=1}^n c_i v_{c,i} + \left( 1 - \sum_{i=1}^n c_i \right) v_w = 0
\]  

(3.29)
Together with equations (3.18) and (3.19) the volume balance forms a system of $N+1$ equations with $N+1$ unknowns. Mathematically the following simple expression can be derived from this system:

$$v_a = \sum_{j=1}^{N} c_j w_{s,j}$$  \hspace{1cm} (3.30)

Substitution of this result in equation (3.18) leads to the following expression:

$$v_{s,i} = \sum_{j=1}^{n} c_j v_{r,j} - v_{z,i}$$  \hspace{1cm} (3.31)

This result was already published by Smith (1966). When the concentration of all fractions is known, the right-hand side of this equation is known as well and the grain velocity can be calculated. Equation (3.31) can be used when the hindered settling relation of a certain grain size is based on the slip velocity, like for instance equation (3.19). The Davis formulae (eq. (3.25)) is directly providing an expression for the velocity of the sediment for a batch sedimentation situation. When this expression is used to compute the slip velocity, the exponent of the factor $(1-c_r)$ must be lowered analogous to equation (3.19). The following expression is obtained:

$$f_i(c_i) = (1-c_i)^{-S_{s,i}} \left( 1 + \sum_{j=1}^{n} (S_{j} - S_{s,i}) c_j \right)$$  \hspace{1cm} (3.32)

### 3.6 Influence of clay fraction

#### 3.6.1 Introduction

In the previous Chapter the influence of the particle size distribution was analysed. The analysis is valid for particles with sizes of the same order of magnitude. A mixture of sand with very fine particles (clay) behaves differently. The fine fraction can be considered as non-settling, but influences the settling of the (relative) coarse particles:

1. The density of the carrier fluid (the mixture of water and clay fraction) is higher than water.
2. The carrier fluid can behave in a non-Newtonian manner at large concentration.

In the following Sections different methods are reviewed to include the influence of the clay fraction on the sedimentation behaviour of sand particles. This contribution must be regarded as a first introduction. The influence is (not yet) included in the hopper sedimentation models presented in this thesis.

In Section 3.6.2 the settling of solid particles in a stagnant fluid is analysed. In Section 3.6.3 the influence of the shear velocity is discussed.
3.6.2 Settling in a stagnant clay suspension

The non-Newtonian behaviour can often be described as Bingham plastic. The shear stress in the fluid is composed of the sum of a yield stress and a shear depended contribution:

$$\tau = \tau_y + \eta \frac{du}{dz}$$  \hspace{1cm} (3.33)

With $\tau_y$ as yield stress and $\eta$ as dynamic viscosity. A method to take the Bingham plastic behaviour into account was proposed by Dedegil, (1987). The basis is the force balance when the terminal velocity is reached:

$$\frac{\pi}{6} (\rho_s - \rho_{supt}) g D^3 - \frac{1}{4} \pi^2 D^3 \tau_y - \frac{\pi}{8} C_D \rho_{supt} D^3 w_0^2 = 0$$  \hspace{1cm} (3.34)

This equation is equivalent to equation (3.1) in stationary settling. Apart from the drag term an extra resistance term caused by the yield stress is present. This contribution is the vertical component of the yield stress integrated over the sphere surface. The density of water is replaced with the density of the clay suspension. Rewriting this equation provides the following expression for the drag coefficient:

$$C_D = \frac{2}{\rho_{supt} w_0^2} \left( \frac{2}{3} (\rho_s - \rho_{supt}) D g - \pi \tau_y \right)$$  \hspace{1cm} (3.35)

The particle Reynolds number for Non-Newtonian fluid is written by Dedegil as:

$$Re_p = \frac{\rho_{supt} w_0^2}{\tau_y + \eta w_0^2}$$  \hspace{1cm} (3.36)

Using this definition it appears that the relation between the drag coefficient and the particle Reynolds number shows almost the same relation as for Newtonian fluids.

$$C_D = \frac{24}{Re_p} \quad Re_p < 8$$

$$C_D = \frac{24}{Re_p} + 0.25 \quad 8 \leq Re_p < 150$$  \hspace{1cm} (3.37)

$$C_D = 0.4 \quad Re_p \geq 150$$

For the laminar regime equations (3.35), (3.36) and (3.37) can be combined to achieve an explicit relation for the settling velocity.

$$w_0 = \frac{g D^2 (\rho_s - \rho_{supt})}{18 \eta} - D \frac{\tau_y}{\eta} \left( 1 + \frac{\pi}{12} \right)$$  \hspace{1cm} (3.38)
In this equation the Stokes settling velocity with a correction for the Bingham Plastic behaviour is recognised. Ansley and Smith (1967) performed a similar analysis, but took into account that a part of the fluid is moving along with the grain. Due to this effect the Reynolds number is written as:

\[
Re_p = \frac{\rho_{\text{susp}} w_0^2}{\eta \frac{w_0}{D} + \frac{7\pi}{24} \tau_b}
\]  

(3.39)

Ansley and Smith plotted the experiments as function of this definition of the Reynolds number versus the drag coefficient (see Figure 3.4). The latter simply defined as for a Newtonian fluid.

\[
C_D = \frac{4}{3} \left( \frac{\rho_s - \rho_{\text{susp}}}{\rho_{\text{susp}} w_0^2} \right) g D
\]

(3.40)

In Figure 3.4 the relation between drag coefficient and Reynolds number defined with equation (3.37) is plotted as well. It is clear that this relation fits the experiments of Ansley & Smith as well, although fitted on a different definition of \( C_D \) and \( Re_p \). Apparently equation (3.37) has a general validity as long as the right combination of \( C_D \) and \( Re_p \) is used.

\[\begin{align*}
\Delta & \quad \text{Exp. Ansley & Smith} \\
C_d &= f(Re) \text{ Bingham} \\
- & \quad C_d = f(Re) \text{ Newton}
\end{align*}\]

\[\text{Figure 3.4 Drag coefficient versus Reynolds number for Bingham Plastic fluid.}\]

3.6.3 Setting in shear flow

When equation (3.35) is evaluated, a special situation occurs when the yield stress exceeds a critical value:

37
\[ \tau_g \geq \alpha_{cr} \left( \rho_s - \rho_{sup} \right) D G \]  

(3.41)

In that case the particle does not settle in a no-flow condition; the shear stresses acting on the particle surface are in equilibrium with the submerged weight as indicated in Figure 3.5 a). From equation (3.35) follows a critical value \( \alpha_{cr} = 2/(3\pi) = 0.21 \). Practical values of \( \alpha_{cr} \) are in a range between 0.048 and 0.2 according to Chhabra, (1993).

![Figure 3.5](image)

**Figure 3.5**  *Shear stress distribution over a particle in stagnant fluid (a) and shear flow (b) (from Talmon et al. 2002).*

In a shear flow the velocity gradient will tend the particles to rotate. The shear stress distribution on the particle surface alters due to this rotational velocity (Figure 3.5 b) resulting in a downward directed component. This downward component leads to a settling behaviour. Talmon et al. (2002) evaluated the force balance indicated in Figure 3.5b for a Herschel-Bulkley model, which is an extension of the Bingham plastic model:

\[ \tau = \tau_s + K \left( \dot{\gamma} \right)^n \]  

(3.42)

Here \( \tau_s \) is the yield stress, \( K \) a viscosity parameter and \( \dot{\gamma} = \partial u / \partial z \) (for horizontal flow) the shear rate of the flow. The following expression for the settling velocity was derived:

\[ w_0 = \alpha \frac{1}{18} \left( \rho_s - \rho_{sup} \right) g D \frac{\partial u}{\partial z} = \alpha \frac{1}{18} \left( \rho_s - \rho_{sup} \right) g D^2 \]  

(3.43)

With: \( \alpha = \) an empirical coefficient and \( \eta = \) apparent viscosity at main flow conditions. The apparent viscosity is defined by:

\[ \eta = \frac{\tau}{\dot{\gamma}} = \frac{\tau_s}{\dot{\gamma}} + K \left( \dot{\gamma} \right)^{n-1} \]  

(3.44)

The mathematical form of this equation is the same as the well-known Stokes formulation even though the flow and stress conditions are quite different.
In Figure 3.5 b the shear stress distribution for a horizontal shear flow is sketched. It is interesting that this distribution is independent of the direction of shear. We can therefore apply equations (3.43) and (3.44) in a 2D flow field using the general expression for the shear rate:

$$\dot{\gamma} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}$$  \hspace{1cm} (3.45)

The validity of the theory is restricted to:

$$\left| \frac{w_0}{D\dot{\gamma}} \right| \leq 1$$  \hspace{1cm} (3.46)

The theory was compared with experiments and the agreement was satisfactory as long as above restriction was fulfilled. On the basis of this analysis it can therefore be concluded that solid particles always settle during flow even when the yield stress exceeds the critical value according to equation (3.41). When the flow is stopped the settling is retarded because of an increase of apparent viscosity (Talmon et al. (2002)).

3.6.4 Final Remark

In the previous sections the impact on the settling velocity of grains is quantified in case the carrier fluid contains a clay fraction that causes non-Newtonian behaviour. These formulations are valid for a very wide range of viscosity. In this thesis emphasis is given to sediments that show a settling behaviour on the time scale of the loading process. The clay content must be quite high (> 10%) before the carrier fluid shows a yield stress. In that case the loaded sediment is practically non-settling and the overflow phase is not present. Therefore the influence of the clay fraction is limited to an increased viscosity and density of the carrier fluid for the area of interest.

3.7 Influence of Turbulence

Grains will generally settle in a turbulent flow field. As already discussed in section 2.3.1 turbulence will influence settling by mixing of the particles in the fluid. This effect is taken into account by the diffusive term in the advection diffusion equation used for suspended sediment transport. However, apart from this mixing effect, the settling velocity of the grains relative to the fluid is also influenced by the turbulent fluctuations. When a particle is subjected to a vertically oscillating fluid the particle will respond to this oscillation. The response will not be completely symmetric and the average vertical velocity of the grain will be affected. This influence is expressed in a variation of the drag coefficient $C_D$ of the grains. Brucato et al (1998) reported that the influence on the drag coefficient could be correlated with the ratio $D/\lambda$. With $\lambda$ as the Kolmogoroff length scale of turbulence, define as:
\[ \lambda = \left( \frac{V^3}{\varepsilon} \right)^{1/4} \]  

(3.47)

Where \( \varepsilon \) is the dissipation rate of turbulent energy. The following relationship was based on experiments (Where \( C_{D,0} \) is the drag coefficient in stagnant flow)

\[ \frac{C_D - C_{D,0}}{C_{D,0}} = 8.76 \cdot 10^{-4} \left( \frac{D}{\lambda} \right)^3 \]  

(3.48)

Because the experimental range for \( D/\lambda \) varied between 3 – 30, a maximum value of 40 was observed for the relative drag coefficient. However, the dissipation rate \( \varepsilon \) during the tests of Brucato et al. (1998) was rather high compared with the situation encountered in a hopper. This resulted in relative high values of \( D/\lambda \).

The experiments were performed with very dilute mixtures, where particles can move around the flow without any restriction. It is questionable if the same large influence can be observed for high concentration. Because of this uncertainty is the influence of the turbulence on settling velocity not included in the 2DV hopper sedimentation model presented in Chapter 7.

### 3.8 Hydrodynamic Dispersion

Although this was not explicitly stated, the settling velocity in the previous part of this Chapter was the time-averaged settling velocity of the grains. The velocities of the individual grains fluctuate relative to this average value. The magnitude of the variation is of the same order as the average velocity. (Ham & Homsy, 1988). The velocity fluctuations are correlated over a distance of order 100 grain diameters (Xue et al. (1992)). This process is analogous with a diffusion process and is called hydrodynamic dispersion or self-diffusion. The diffusion coefficient is the product of a length and velocity scale. In the literature concerning this subject the grain diameter \( D \) and settling velocity \( w_0 \) are commonly used to normalise the self-diffusion.

In Figure 3.6 the dimensionless diffusion coefficient is plotted versus the concentration using data from Nicolai et al. (1995) and Segrè (2001).
Figure 3.6  Normalised self-diffusion coefficient versus concentration.

The settling velocity was in the Stokes regime for these experiments. The normalised diffusion coefficient decreases with concentration due to the fact that less space is available at higher concentration. When the diffusion is normalised with the actual settling velocity \( w_s \) (including hindered settling) the diffusion coefficient is more or less constant up to a concentration of 0.4 and decreases at higher values. The order of magnitude of the self-diffusion is maximal 10\( D_w \). This leads to values of about \( 10^{-5} \) m\(^2\)/s for 100-200 \( \mu \)m sand. This might be of importance in case of purely one-dimensional sedimentation, but is very small compared with the diffusion due to turbulent mixing when the sand-water mixture is flowing like in a hopper as will be shown in Chapter 7.

3.9 Hindered settling versus Permeability.

When the concentration is reaching values close to those encountered in a settled bed very little space is available for grains to move around. Self-diffusion and hence segregation is suppressed. In that case the ability for water to flow out in vertical direction determines the settling velocity of a mixture. It will be shown that using the permeability of the mixture and Darcy’s law the settling velocities can be predicted as well. The permeability of sand in the laminar regime can be computed using the equation of Den Adel (1987):

\[
k = \frac{g}{160\nu} D_{is} \left( \frac{1-c}{c} \right)^3 \frac{1}{c^2}
\]  
\( (3.49) \)
Darcy's law reads:

\[ u = -k \frac{dh}{dx} \]  

(3.50)

Where \( u \) is the filter velocity (or superficial velocity in chemical literature) and \( dh/dx \) the pore pressure gradient. The superficial velocity is relative to the grains. In one-dimensional sedimentation:

\[ u = (1+c)w_i \]  

(3.51)

The excess pore pressure equals the submerged weight of the suspended grains and reads:

\[ \frac{dh}{dx} = \frac{\rho_m - \rho_w}{\rho_w} = c\Delta \]  

(3.52)

Combination of these four equations and solving for \( w_i \) leads to the following expression:

\[ w_i = \frac{\Delta gD_{15}^2}{18\nu} \frac{18(1-c)^3}{160c(1+c)} \]  

(3.53)

The settling velocity can hence be written as the product of the Stoke settling velocity of a single grain and a hindered settling function. The settling velocity computed with equation (3.53) is compared with the settling velocity computed with equation (3.4) and (3.5) for two different (uniform) grain sizes in Figure 3.7. It is shown that both approaches yield similar results.

![Figure 3.7](image)

**Figure 3.7**  Settling velocity versus concentration.
3.10 The influence of air bubbles on the settling velocity of sand

3.10.1 Introduction

Loading pipes or diffusers are very often located above the overflow level onboard hopper dredgers. As a consequence the incoming mixture plunges into the hopper. When the impact velocity of the impinging mixture into the hopper exceeds a certain critical velocity, air bubbles are entrained. The downward directed flow in the impingement section drags a certain fraction of bubbles to the bed. In the remaining part of the hopper the direction of the flow is horizontal or upward hence the majority of the bubbles eventually escape to the surface. Only a very small percentage is captured in the pore water in the bed. As a consequence the settling particles in the hopper will encounter bubbles moving in the opposite direction and possibly the settling is affected by this interaction.

The following paragraphs only provide a short introduction of the possible mechanisms, which can affect settling, and a brief overview of the investigations performed in the framework of the current research programme. The preliminary results were not (yet) implemented in the hopper sedimentation models, which are presented later in this thesis.

Overview

The presence of air bubbles can affect the sedimentation in several ways. To quantify this influence the following factors must be taken into account.

- The volume of air entrained by the plunging mixture
- The bubble size distribution of the entrained air
- The possible influence of air bubbles on sedimentation

3.10.2 Air entrainment rate

The problem of air entrainment in hydraulic structures (weirs, siphons etc.) has been studied intensively in the past. The air entrainment rate depends on the impact velocity of the plunging jet, jet dimensions (width and thickness) and fall height (from the discharge structure to the hopper water surface. Empirical relations have been formulated based on experiments using clear water (Ervine, (1976) Bin, (1988)). In general these formulas cannot be applied to the situation present in a hopper due to the difference in geometry. Ervine (1998) made a review on air entrainment in hydraulic structures. One of the main conclusions was that the velocity of the impinging jet was the most important factor. The jet thickness did not have any influence as long as the value exceeded approximately 30 mm. This can be explained by the fact that the entrainment takes place at the interface between the impinging jet and stagnant water. A regression analysis on a number of experimental results resulted in the following expression for the air entrainment rate:
\[ q_a = 2 \times 10^{-5} (u_i - 1)^2 + 3 \times 10^{-4} (u_i - 1)^2 + 0.0074 (u_i - 1) - 0.0058 \] (3.54)

With \( q_a \) as the entrainment rate per unit width and \( u_i \) the jet velocity.

### 3.10.3 Bubble size distribution

In Figure 3.8 a plunging jet with discharge \( q_a \) into stagnant water is sketched. At the location where the jet enters the water surface air is entrained (\( q_L \) is entrained volume per unit time). Below a plunging jet two areas can be distinguished:

![Figure 3.8 Entrainment of air bubbles below plunging jet.](image)

1. The cone-shaped area below the impingement point (point where jet enters water surface) where, owing to the large turbulence level, small bubbles are created. These bubbles are called the primary bubbles.
2. Around this area large secondary bubbles are rising. These bubbles are formed by coalescence of the primary bubbles. The coalescence is only possible in areas where the velocity gradients are small enough and the fluid properties permit this process. It is known that in salt water electrostatic forces hinder coalescence.

Very little has been reported on the bubble size distribution resulting from plunging jets. Ervine (1998) reports that the bubble size decreases as the impact velocity increases. The (very) approximate relation is:

\[ d_b = 0.03/u_i^2 \] (3.55)

To achieve more insight on this subject, and to investigate the influence of the salinity of the fluid on the bubbles size distribution, experiments were carried out at GeoDelft within the framework of this research program (Lubking et al., 2000).

Experiments were carried out with an axisymmetric jet plunging in fresh and salt water. The fall height of the jet, impact velocity and nozzle diameter were varied. It appeared that the results in fresh water were clearly different from salt water.

In fresh water the influence of velocity, jet diameter and fall height was very small on the bubble size distribution. The bubble size distribution for all tests on fresh water
could be fitted with a normal distribution with an average bubble diameter of 3 mm and a standard deviation of 1 mm. In salt water (15 g NaCl/l) the bubble size was much smaller than in fresh water and decreased with the impact velocity. The bubble size ($D_{50}$) varied between 0.4 – 0.75 mm depending on the velocity. The largest bubbles were around 3 mm.

The large difference between fresh and salt water results from the difference in coalescence. It is expected that both in fresh as in salt water the primary bubbles decrease with impact velocity. In fresh water these bubbles coalesce to bubbles of approximately 3 mm. In salt water the coalescence is hindered owing to the fact that the bubbles are charged electrically and repulse each other.

3.10.4 Influence on sedimentation

The air bubbles can influence the sedimentation process in different ways. The different mechanisms all have a distinct length scale.

Macro scale

The presence of the air bubbles in the plunging jet decrease the over all (air-water-sediment) density of the mixture. The negative buoyancy of the mixture in the inflow section is therefore reduced. This can lead to a reduction of the acceleration of the mixture in the inflow section.

Micro scale

On the scale of the bubble size different mechanisms can be responsible for bubble-grain interaction (see Figure 3.9).

![Figure 3.9 Interaction between air bubble and grains.](image-url)
1 Turbulent wake below rising bubble.

In fresh the water average bubble size is 3 mm. The rising velocity of a bubble with that size is approximately 0.3 m/s, hence the particle Reynolds number is around $10^4$. The return flow induced by a rising bubble is therefore turbulent in most cases. The eddies developing in the wake of the bubble increase the turbulence level in the mixture. Beside this effect it is possible that grains are trapped in this wake and transported vertically over some distance with the bubble.

2 Attachment of grain to the bubble.

When a grain collides with a bubble it is possible that the particle sticks to the bubble-fluid interface and is transported to the surface with the bubble. On the surface a foam layer develops which contains the entrapped particles. On board a hopper dredger this foam layer is very often visible but the amount of sand in this layer is negligible. Clay particles are present but the volume present in the foam layer is negligible compared with the total influx of these particles. This mechanism does not play any role of importance.

3 Acceleration of flow beside bubble

The rising bubble creates a return flow beside the bubble. A falling grain is affected by this flow. The grain has to travel over a longer distance, and is accelerated in downward direction. This acceleration does not compensate the longer distance, so the over all effect is a decreased settling velocity.

The behaviour of gas bubbles in bubble columns has been investigated intensively. On the behaviour on three phase systems (gas – fluid – solid) for high concentration of solid particles literature can be found as well, but in most cases the influence of the fluid – solid phases on the rise velocity on a bubble is investigated (see Darton & Harrison, (1974), Tsuchiya, (2001)) and not the influence of the gas on the settling velocity of the solid particles.

It was therefore decided to investigate the possible influence of air on the settling velocity of sand in the framework of this research programme. The first step is to study the influence at low solids concentration. For this purpose experiments were carried out using a sedimentation column equipped with a gas distributor at the bottom of the column. The height of the column was 2.85 m. At the top a sample of sand could be released in the fluid. At the bottom, below the gas distributor, a weighing device was installed. From the recorded weight increase in time the settling velocity distribution can be computed (Van Rhee et al. (2002)). In fact not only the settling velocity, but the vertical transport of sand, so diffusion in the column is included.
Figure 3.10 Influence of air on settling velocity.

Figure 3.10 shows a result from one of the test series performed on fresh water. The cumulative settling velocity distribution for uniform glass beads is plotted for different air concentrations \( \varepsilon \). The air (volume) concentration was measured by the difference in water level after air was injected in the water column. A very narrow grain size distribution was used for the solid particles (all grains between 180 and 212 \( \mu \text{m} \)). The theoretical settling velocity for the beads is 2.2 cm/s for 180 \( \mu \text{m} \) and 2.8 cm/s for 212 \( \mu \text{m} \), which is lower than measured. Owing to the influence of the air the settling velocity becomes wider distributed. This phenomenon is caused by diffusion from turbulence generated by the rising bubbles (Van Rhee et al. (2002)). Apart from this effect a slight increase of the average (50\% value) of the (effective) settling velocity can be observed.

It can be concluded that for low solid concentration the influence of the air bubbles on micro scale on the settling velocity (apart from the mixing effect) is rather small. The reduction of the settling velocity as described in Section 3.7 was not observed in the experimental results, which supports the approach not to include this phenomenon in the hopper sedimentation model. It is not clear if the small influence of the bubbles on settling is also valid for large solid concentration; this has to be investigated in the future. The influence of the bubbles on macro scale (buoyancy effect) has not been quantified either. Owing to these uncertainties the above findings are not included in the models presented in Chapter 6 and Chapter 7. More research is needed for this subject in the future.
3.11 Sedimentation velocity

In this Chapter attention was focussed so far on the settling velocity of solid particles and how this quantity is affected by different influences. Very often the term sedimentation velocity is used as a synonym for settling velocity. In this thesis the sedimentation velocity is defined as the vertical velocity of the bed-water interface. In this section the relation between the sedimentation velocity, settling velocity and near bed concentration is discussed. An expression for the sedimentation velocity can be derived by application of the continuity equation for particles near the bed. In Figure 3.11 a control volume is shown. The bed-water interface intersects this volume. The volume of grains in the bed and suspension is $V_b$ and $V_s$ respectively.

![Figure 3.11 Control volume near settled bed.](image)

For convenience the horizontal gradient in sediment transport is zero. The volume balance reads:

$$\frac{d}{dt} (V_b(1-n_b) + V_s c_b) = S = w_i c_b$$  \hspace{1cm} (3.56)

Where $n_b$ is the porosity of the bed, $c_b$ the near bed concentration (just above the bed) and $v_{sed}$ the sedimentation velocity. This equation can be simplified to:

$$v_{sed} (1-n_b) - v_{sed} c_b + V_b \frac{dc_b}{dt} = w_i c_b$$  \hspace{1cm} (3.57)

A small control volume is analysed, so the third term in the left-hand side of this last equation can be neglected. Equation (3.57) leads to the following relationship for the sedimentation velocity (using Richardson & Zaki to include the effect of hindered settling):

$$v_{sed} = \frac{w_i c_b}{1-n_b - c_b} = \frac{w_i c_b (1-c_b)^7}{1-n_b - c_b}$$  \hspace{1cm} (3.58)

This equation is only valid in the absence of erosion, so for sedimentation is stagnant flow conditions.

Note that the near bed concentration $c_b$ appears both in the numerator as the denominator of this expression. In literature the expression for the sedimentation
velocity is often written without \( c_b \) in the denominator, which is only permitted for low near bed concentration. In Figure 3.12 the different relationships are compared (for 100 \( \mu \)m sand). The expression with \( 1 - n_0 \) in the denominator shows an unrealistic optimum sedimentation velocity for \( c_b \approx 0.2 \). For high concentration a large error in the computed sedimentation velocity is made in case this relationship is used.

\[
\begin{align*}
\nu_{\text{sed}} &= c_b \frac{W_s}{(1 - n_0 - c_b)} \\
\nu_{\text{sed}} &= c_b \frac{W_s}{(1 - n_0)}
\end{align*}
\]

Figure 3.12 Sedimentation velocity versus near bed concentration. Influence of the concentration term in the denominator

In Figure 3.13 the sedimentation velocity is plotted versus the near bed concentration for four different sands.

\[
\begin{align*}
\text{D=100} & \quad \text{140} & \quad \text{180} & \quad \text{220 \( \mu \)m}
\end{align*}
\]

Figure 3.13 Sedimentation velocity versus near bed concentration for different sands.
The exponent $n$ is according to the original values proposed by Richardson & Zaki. The settling velocity $w_0$ is computed using equations (3.4) and (3.5). The porosity $n_0$ is 0.4. It is remarkable that for fine sands the sedimentation velocity is not very sensitive of $c_h$ over a broad range. This phenomenon is caused by the large value of the hindered settling exponent $n$ for small particle Reynolds numbers (see Figure 3.3). In non-stagnant flow conditions, where erosion might be important, the sedimentation velocity is computed using the difference between the settling flux $S$ and the erosion (or pickup) flux $E$:

$$v_{sed} = \frac{S - E}{(1 - n_0 - c_h) \rho_s}$$  \hspace{1cm} (3.59)

The settling flux equals:

$$S = \rho_s c_h w_s$$  \hspace{1cm} (3.60)

The pickup rate $E$ is determined using an empirical relation. These relations are however based on experiments with low sediment concentration (see Van Rijn, 1984a). More on the influence of non-stagnant flow conditions on the sedimentation velocity for high concentration can be found in Chapter 5.
Chapter 4

Model hopper sedimentation tests

4.1 Overview of experimental programs

A large part of the present research programme consisted of laboratory experiments. The following investigations were carried out:

One-dimensional sedimentation tests

In a sedimentation column the sedimentation of sand was investigated in a one-dimensional situation. Initial concentration, particle size distribution (special attention was given to graded sand) and level of turbulence were varied. Some of the results and a brief description of the experimental arrangement can be found in Chapter 6, together with the analysis of the 1DV model. For more detailed information reference is made to Runge & Ruig, (1998) and Klerk & Meulepas, (1998) and Van Rhee (2001).

Influence of air bubbles on the settling velocity of sand

Experiments were carried out to investigate the bubble size distribution of air entrained by a plunging jet. The fall height of the jet, jet diameter, impact velocity and salinity of the fluid was varied (Lubking et al. 2000).

One-dimensional tests were carried out in a sedimentation column to investigate the influence of air bubbles on the sedimentation of sand at low concentration. The influence of variation in the particle size distribution, air concentration and salinity were examined. The experimental set-up and the conclusions drawn from these experiments are summarised in Section. 3.10. For more information reference is made to van Spengen (2001) and Van Rhee, d’Angremond et al. (2002).

Large scale hopper sedimentation tests

Model hopper sedimentation tests with large dimensions (to minimise scale effects) were performed (Van Rhee, 1999, 2001a). During these experiments the inflow and outflow quantities were recorded but the attention was focussed on the situation inside the hopper. Numerous instruments were installed in the hopper to measure the velocity and concentration distribution as a function of time and space. In this Chapter the experimental set-up, test program, instrumentation and the analysis of the results are discussed together with a short introduction to the field of scaling rules.
Closed flume sedimentation tests
To investigate and quantify the erosion-sedimentation behaviour in the density current close to the bed in full scale circumstances sedimentation tests were performed in closed flumes (Van Rhee, 1999, Van Rhee & Talmon, 2000). These experiments are reviewed in Chapter 5.

4.2 Motivation of model hopper sedimentation tests
In the past, the dredging contractors generally carried out model hopper sedimentation tests (mostly "in house"). The dimensions of the test facilities were often rather small and the measurements were limited to recording the inflow and outflow quantities. The processes inside the hopper were more or less regarded as a "black box". The model hopper sedimentation tests carried out in the framework of the present research programme had the following objectives:
- To gain more insight into the physical process inside the hopper.
- To quantify the influence of process parameters
- To get high quality test data for model validation.

To achieve these goals it was important to minimise scale effects. Therefore the tests were preceded by an analysis of the scaling rules to determine the minimum dimension of the test arrangement and to quantify the relationship between model and prototype values.

4.3 Scaling Rules
The dimensions of the geometry of the experiments are smaller than the dimensions of an actual hopper. In order to scale the results to prototype values and to choose the right values for the parameters (discharge, sand etc.) it is important to know the relationship between the values in model and prototype (true scale). These can be quantified using scaling rules. A scale factor for a certain property A is defined as:

\[
n_A = \frac{A_{\text{prototype}}}{A_{\text{model}}} \tag{4.1}
\]

Geometry
The length L, width B and depth H in a hopper are of the same order of magnitude. The experiments have to be performed on a non-distorted scale:

\[
n_L = n_B = n_H \tag{4.2}
\]

Time scale
The scale for loading time follows from continuity. When \( \bar{c}_w \) and \( \bar{c}_o \) are the time averaged inflow and outflow volumetric concentration, \( Q \) is the discharge in the hopper
(assumed to be constant for simplicity). $V$ the volume of sediment settled in the hopper and $n_0$ the porosity of the bed, the volume balance for the solids in the hopper reads:

$$t_L Q \left( c_{in} - c_r \right) = (1 - n_0) V$$  \hspace{1cm} (4.3)$$

Where $t_L$ is the loading time. The overflow flux $OV_{flux}$ is defined as:

$$OV_{flux} = \frac{Q_c c_r}{Q_{in} c_{in}}$$  \hspace{1cm} (4.4)$$

Using this definition, equation (4.3) can be written as:

$$t_L = \frac{(1 - n_0) V}{Q_{in} c_{in} (1 - OV_{flux})}$$  \hspace{1cm} (4.5)$$

If the overflow loss is properly scaled and the porosity of the bed is the same in model and prototype, the following rule applies (using $n_Q = n_u n_L^2$):

$$n_r = n_q n_0^{-1} n_c^{-1} = n_L n_u^{-1} n_c^{-1}$$  \hspace{1cm} (4.6)$$

**Overflow rate**

We have seen in Chapter 2 that the overflow rate (equation (2.4)) is an important variable. The overflow rate can be made dimensionless with the settling velocity. The resulting parameter is the dimensionless overflow rate $H^*$ (Wijnant, 1993). The parameter $H^*$ is:

$$H^* = \frac{Q}{BL \omega_s}$$  \hspace{1cm} (4.7)$$

This parameter is in fact the reciprocal value of the grain settling efficiency in the Camp model (equation (2.3)). The dimensionless overflow rate in model and prototype must be the same, hence $H^*$ is on the right scale when:

$$n_{H^*} = 1$$  \hspace{1cm} (4.8)$$

Using the definition of $H^*$, this rule can be written as:

$$n_Q = n_u n_L^2 = n_w n_L^2 \Rightarrow n_u = n_w$$  \hspace{1cm} (4.9)$$

The scales of flow and settling velocity must be the same.

**Vertical mixing**

The scaling rule for the overflow rate can be regarded as a scale for the advection of sediment in the hopper. The other important sediment transport mechanism is diffusion by turbulence. It is known that turbulence is affected by density gradients (stratification). The influence of stratification on mixing can be quantified using the flux Richardson number defined as the ratio of the buoyancy dissipation and production terms of turbulence:
\[ Ri_j = -\frac{\Delta g c'w'}{\rho u'w' \frac{\partial u}{\partial z}} \] (4.10)

Where a prime denotes the fluctuation part of the horizontal and vertical components \( u \) and \( w \) and of the concentration \( c \). The overbar denotes averaging over the turbulent time scale (Turner, 1973). The terms with the time averaged fluctuations can be expressed in gradients of velocity and concentrations using the eddy-viscosity concept of Boussinesq (Rodi, 1993):

\[ \overline{u'w'} = V_e \frac{\partial u}{\partial z} \]

\[ \overline{c'w'} = \frac{V_e}{\sigma} \frac{\partial c}{\partial z} \] (4.11)

Where \( \sigma \) is the Schmidt-Prandtl number. Substitution of (4.11) in (4.10) leads to the following expression for the flux Richardson number:

\[ Ri_j = -\frac{\frac{\Delta g c}{\partial z}}{\frac{\partial u}{\partial z}} = \sigma^{-1} Ri \] (4.12)

Where \( Ri \) is the gradient Richardson number. It is important that the ratio between the generation and dissipation of turbulence in the model is the same as in the prototype. Hence the scale for the flux Richardson number must be unity.

\[ n_{ri} = 1 \quad \Rightarrow \quad \frac{n_p}{n_l} = \frac{n_p}{n^2_l} \quad \Rightarrow \quad n_u = \sqrt{n_l} \] (4.13)

**Density Currents**

Another important dimensionless parameter, when density currents are present, is the internal Froude number:

\[ F_i = \frac{u}{\sqrt{\frac{\rho_m - \rho_w}{\rho_m} gh}} \] (4.14)

Again the scale factor must be unity for this quantity, which is only in accordance with the outcome of the Richardson number requirement, when the concentration differences are equal in model and prototype:

\[ n_{ci} = 1 \] (4.15)
4.3.1 Erosion versus Sedimentation

The sedimentation velocity follows from (Section 3.11):

\[ v_{sed} = \frac{S - E}{(1 - n_0 - c_b) \rho}, \]  

(4.16)

The sedimentation velocity is of the right scale when the scale of the settling flux \( S \) equals the erosion (Pickup) flux \( E \). Since the concentration scale is unity and the settling flux is proportional to the product of the settling velocity and the near bed concentration \( c_b \) (equation (3.60)), the scale for the erosion reads:

\[ n_E = n_w. \]  

(4.17)

To derive a scaling rule for the erosion is not as straightforward as for the other quantities discussed above since different empirical relations are available and none of these were derived for the high concentration present in a hopper. Most expressions have in common that the pickup rate \( E \) is a function of the difference between the Shields parameter and a critical Shields parameter.

\[ E :: \rho_s \sqrt{\Delta g D} (\theta - \theta_c)^\alpha \]  

(4.18)

The Shields parameter is a dimensionless form of \( u \).

\[ \theta = \frac{u_s^2}{\Delta g D} = \frac{f}{8} \frac{\bar{u}^2}{\Delta g D} \]  

(4.19)

The critical value of the Shields parameter determines the start of movement (transportation) of a sand grain due to currents and depends on the diameter of the grains. The scaling rule for equation (4.18) can be written as:

\[ n_E = n_\Delta^{0.5} n_D^{0.5} n_0^{\alpha \theta / (\theta - \theta_c)} \]  

(4.20)

When the Shields parameter \( \theta \) is large relative to the critical value \( \theta_c \), using (4.9) and (4.19), this rule can be written as (\( n_\Delta = 1 \) \( n_f = 1 \)):

\[ n_E = n_\alpha^{2\alpha} n_D^{0.5 - \alpha} = n_w^{2\alpha} n_D^{0.5 - \alpha} \]  

(4.21)

This rule is equal to equation (4.17) in case \( \alpha = 0.5 \). This value is however low compared to the values published. In the pickup function of Van Rijn (1984) \( \alpha = 1.5 \) is used (equation (2.25)). The scale for pickup rate \( E \) therefore differs from the scale for settling flux \( S \). This is called a scale effect. The influence of this effect is illustrated below with an example.

Suppose that model tests are executed to investigate the hopper sedimentation process for 200 \( \mu m \) prototype sand. For different length scales, the scales for settling velocity and pickup rate can be computed using (4.9), (4.13) and (4.21). The results of this procedure are shown in Table 4.1.
<table>
<thead>
<tr>
<th>$n_L$</th>
<th>$n_m$</th>
<th>$D_m$ [μm]</th>
<th>$n_P$</th>
<th>$n_E$ ($\alpha = 0.5$)</th>
<th>$n_E$ ($\alpha = 1.0$)</th>
<th>$n_E$ ($\alpha = 1.5$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>[-]</td>
<td>[-]</td>
<td>[-]</td>
<td>[-]</td>
<td>[-]</td>
<td>[-]</td>
<td>[-]</td>
</tr>
<tr>
<td>2</td>
<td>1.41</td>
<td>163</td>
<td>1.23</td>
<td>1.41</td>
<td>1.81</td>
<td>2.31</td>
</tr>
<tr>
<td>3</td>
<td>1.73</td>
<td>145</td>
<td>1.38</td>
<td>1.73</td>
<td>2.55</td>
<td>3.77</td>
</tr>
<tr>
<td>4</td>
<td>2.00</td>
<td>134</td>
<td>1.49</td>
<td>2.00</td>
<td>3.27</td>
<td>5.36</td>
</tr>
<tr>
<td>5</td>
<td>2.24</td>
<td>127</td>
<td>1.57</td>
<td>2.24</td>
<td>3.98</td>
<td>7.10</td>
</tr>
<tr>
<td>6</td>
<td>2.45</td>
<td>120</td>
<td>1.67</td>
<td>2.45</td>
<td>4.65</td>
<td>8.82</td>
</tr>
<tr>
<td>7</td>
<td>2.65</td>
<td>115</td>
<td>1.74</td>
<td>2.65</td>
<td>5.31</td>
<td>10.65</td>
</tr>
</tbody>
</table>

Table 4.1: Comparison between the scales for sedimentation and pickup rate.

The third column shown the particle size to be used in the model corresponding to the length scale given in the first column. Column 5-7 show the scale for the pickup rate for different values of $\alpha$. For values of $\alpha$ larger than 0.5 (which is to be expected) the scale for the pickup rate is larger than the scale for the settling velocity, which implies that erosion is underestimated in a model test.

One should therefore always be very careful to scale the results of model tests directly to prototype values. To minimise this scale effect, the difference between the particle size of model and prototype should be as small as possible. Laboratory tests should be executed on the smallest length scale (largest model dimensions).

4.3.2 Application of the scaling rules

The scale of the laboratory experiments can only be determined when the size of the prototype is defined. In Table 2.1 it can be seen that trailing suction hopper dredgers are available in a wide range of sizes. The smaller ships can in fact be regarded as a model of the larger ones. Hence the choice is not straightforward. Therefore a more practical approach was followed.

The experiments should be executed on the largest size possible to minimise scale effects. The results from the laboratory experiments can be translated to the different prototype dimensions using the scaling rules from the previous section.

Possible restrictions on the model size are a consequence of the limitations of the existing flumes were the tests had to be executed:

1. Maximum available width.
2. Maximum available depth.
3. Maximum discharge of the available pumps.

In most cases the depth or discharge appeared to be definitive. Another important issue is the selection of the model sand. When the model sediment is too coarse no overflow
losses develop and the results of the tests are not meaningful. On the other hand the sand cannot be too fine either to avoid cohesive behaviour.

Suppose for instance that model tests are executed to study the loading process of 150 μm sand in a 18,000 m³ TSHD. Table 2.1 shows that the length of a hopper of that size is about 66 m. Suppose that the length of the model hopper is 4 m. This implies that the length scale is \( n_L = 16.5 \). The scale for the settling velocity follows from (4.9) and (4.13): \( n_w = n_s = \sqrt{n_L} = 4.1 \). To satisfy this rule 65 μm model sediment should be used, which behaves quite differently when compared with the 150 μm prototype sand.

The model dimensions were based on the following restrictions and assumptions.

1. The available depth was restricted to approximately 2 m.
2. A practical depth/width ratio of the model hopper is 1.5.
3. A practical Length/Width ratio of the model hopper is 4.
4. Maximum available model pump discharge was 0.14 m³/s
5. The minimum grain size of the model sand (based on practical and economical considerations) was 100 μm.
6. The value of the dimensionless overflow rate must be high enough to achieve overflow losses in the model to avoid meaningless results. From previous tests (Wijnant, 1993) it was concluded that the value of \( H^* \) should be varied up to 2.5 - 3.

Application of restriction 1., 2. and 3. yields a model hopper with a length of 12 m, width of 3 m and depth of 2 m. The next step is to check if the available pump capacity (discharge \( Q_m \)) is sufficient. The relation between discharge and the dimensionless overflow rate is:

\[
Q_m = H^* B L w_s
\]  
(4.22)

A dimensionless overflow rate \( H^* = 2.5 \) is used. The settling velocity of 100 μm model sand using a near bed concentration of \( c_b = 0.2 \), including the effect of hindered settling is 1.4 mm/s. Substitution of these values in (4.22) yields: \( Q_m = 0.125 \) m³/s, hence the available capacity is sufficient (restriction no. 4).

Now that the model dimensions have been established, scaling rules can be applied to determine the corresponding values for discharge and grain size on prototype scale. The first important step is the determination of the length scale. This is simple when only one prototype hopper is analysed. The length/width ratio in the model is chosen equal to this value in prototype. The ratio between the prototype and model hopper length is taken for the length scale \( n_L \). When the model results are applied to more than one hopper a problem arises because length/width ratios differ (see Table 2.1), and equation (4.2) does not satisfy all prototype hoppers.
In Section 2.3 the importance of the overflow rate was highlighted. In this parameter the surface area of the hopper includes the influence of geometry. It is therefore proposed to base the length scale on the surface area:

\[ n_L = \sqrt{n_A} = \sqrt{\frac{(BL)_{\text{prototype}}}{(BL)_{\text{model}}}} \]  

(4.23)

Where \( n_A \) is the scale factor for the hopper surface area.

The results of this analysis are shown in Table 4.2. In this Table seven columns are shown with the following values:

1. Name of the hopper dredger
2. Volume of the hopper
3. Length of the hopper
4. Length scale based on a model length of 12 m.
5. The settling velocity in prototype. This value is found applying equation (4.13) and (4.9): \( n_w = \sqrt{n_L} \)
6. The value of the \( D_{50} \) in prototype corresponding to the settling velocity of column 5 using a near bed concentration of \( c_b = 0.2 \) (the same value as in the model, since equation (4.15) applies \( n_c = 1 \))
7. \( Q_{pm} \) is the discharge in prototype corresponding to a variation of the model discharge between 0.075 - 0.140 m³/s using \( n_Q = n_w n_L^2 = n_L^{2.5} \).
8. \( Q_p \) is the actual prototype discharge of the hopper.

<table>
<thead>
<tr>
<th>Ship</th>
<th>Volume [m³]</th>
<th>Length [m]</th>
<th>( n_L )</th>
<th>( w_s ) [mm/s]</th>
<th>( D_{50} ) [μm]</th>
<th>( Q_{pm} ) [m³/s]</th>
<th>( Q_p ) [m³/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fairway</td>
<td>23490</td>
<td>87.0</td>
<td>7.25</td>
<td>3.8</td>
<td>161</td>
<td>9.8-18.3</td>
<td>13.6</td>
</tr>
<tr>
<td>Amsterdam</td>
<td>18000</td>
<td>66.0</td>
<td>5.50</td>
<td>3.3</td>
<td>154</td>
<td>5.9-11.1</td>
<td>11.4</td>
</tr>
<tr>
<td>HAM 310</td>
<td>8225</td>
<td>46.3</td>
<td>3.86</td>
<td>2.8</td>
<td>145</td>
<td>3.1-5.9</td>
<td>11.4</td>
</tr>
<tr>
<td>Volvox Delta</td>
<td>8143</td>
<td>54.6</td>
<td>4.55</td>
<td>3.0</td>
<td>145</td>
<td>3.4-6.4</td>
<td>9.4</td>
</tr>
<tr>
<td>Cornelia</td>
<td>6395</td>
<td>52.4</td>
<td>4.37</td>
<td>3.0</td>
<td>143</td>
<td>2.9-5.3</td>
<td>7.6</td>
</tr>
<tr>
<td>HAM 311</td>
<td>3522</td>
<td>44.0</td>
<td>3.67</td>
<td>2.7</td>
<td>140</td>
<td>2.0-3.8</td>
<td>3.8</td>
</tr>
<tr>
<td>Heron</td>
<td>974</td>
<td>34.3</td>
<td>2.86</td>
<td>2.4</td>
<td>128</td>
<td>0.9-1.7</td>
<td>1.0</td>
</tr>
</tbody>
</table>

| Table 4.2 | Translation from model values to prototype. |

Example:

In the model tests 100 μm sand is used. The model results are equivalent to results on 153 μm sand on prototype scale when the “Amsterdam” is used as prototype (using a discharge in prototype between 5.0 - 11.1 m³/s). It appears that the actual prototype
value for the discharge $Q_p$ is close to the upper limit of the scaled model values. The actual value for the discharge is between the scaled limit for about 50% of the hoppers of above Table. For some ships the actual discharge is well above the model range. Pump capacity should be enlarged or the model dimension should be smaller. It was however decided not to reduce the size of the arrangement to suit all hoppers mentioned in Table 4.2, but to perform the tests on the largest model dimensions possible.

4.4 Test arrangement

The experiments were carried out in the dredging flume of WL Delft Hydraulics. A schematic overview of the arrangement is given in Figure 4.1 and Figure 4.2.

![Diagram of experimental set-up](image)

**Figure 4.1**  *Top view of experimental set-up.*

The set-up comprised four basins: a sand storage basin that contained 150 m$^3$ of sand, a mixing tank, the model hopper and a secondary settling basin in which the sand fractions from the overflow mixture could settle. During a test a sand water mixture was sucked from the storage basin and pumped to the mixing tank that buffered possible flow variation and damped concentration variation of the suction process. From the mixing tank the mixture flowed into the inflow construction located in the model hopper. The discharge to the hopper was controlled using a control valve. The overflow mixture flowed to a container in which two submersible pumps were placed. The two pumps discharged the overflow mixture into the secondary settlement basin in which the sand fraction present in this mixture could settle. The excess water from the secondary settling basin flowed under gravity back to the storage basin. The secondary settling basin was an essential component in the arrangement. Without this basin the
overflow mixture, containing the finer fractions, would flow directly back to the storage basin leading to a changing particle size distribution during the test. The model hopper had a length of 12 m, a width of 3.08 m and a height of 2.5 m. The maximum overflow level was situated at 2.25 m from the bottom, so the hopper contained approximately 83 m$^3$ when fully filled. In the storage basin a surplus of sand was placed to compensate for the losses during the loading process. One of the side walls of the hopper contained glass sections, which made visual inspection of the process possible. On the bottom of the storage basin fluidisation pipes were installed, which enabled a large suction production.

![Diagram](image)

**Figure 4.2** Side view of experimental set-up.

After completion of a test the model hopper was full of sand. The amount of (fine) sand in the secondary settlement basin was dependent on the magnitude of the overflow loss and could be substantial. Before the next experiment could be commenced sand was removed from the hopper and the secondary settling basin and transported to the sand storage basin. This activity was time consuming therefore only one test per day could be performed.

### 4.5 Test Program

In total 19 experiments were carried out. The following parameters were varied:

- Discharge $Q_a$.
- Inflow mixture density $\rho_m$.
- Inflow configuration. The following three structures were used:
  - A diffuser (2D), distributing the mixture over the total width of hopper.
  - A vertical pipe (Pipe), discharging the mixture in one of the corners of the hopper.
  - A spraying pipe (see Figure 2.3) (Spray pipe), distributing the mixture over the total width of the hopper.
Overflow system. The following two structures were used:
- Edge over total width of hopper (2D).
- Round overflow (see Figure 2.4) (Round).

- Overflow level $h_o$.
- Water level in the hopper at start of a test $h_{start}$.
- Particle size distribution of the sediment. The particle size distributions are shown in Figure 4.3. Samples were taken from the inflow during every test. All samples were used to compute the density. The particle size distribution was determined from approximately half the number of all samples collected. Every line in the figure below represents the average cumulative particle size distribution of the measured particle size distributions. The indices in the graph indicate the test number. However, the variation of the PSD within one type of sand is small, hence the difference between tests cannot be obtained from this figure. The values of $D_{10}$, $D_{50}$ and $D_{60}$ are shown in Figure 4.4.

![Graph showing particle size distributions](image)

*Figure 4.3*  Particle size distributions of all tests taken from the inflow.
### Figure 4.4 Grain diameter $D_{10}$, $D_{50}$, and $D_{60}$ for all tests (inflow).

<table>
<thead>
<tr>
<th>Test</th>
<th>$Q_{in}$ [m$^3$/s]</th>
<th>$\rho_{in}$ [kg/m$^3$]</th>
<th>$D_{50}$ [μm]</th>
<th>Inflow Config.</th>
<th>Overfl. Config.</th>
<th>$h_{ov}$ [m]</th>
<th>$h_{start}$ [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.099</td>
<td>1310</td>
<td>140</td>
<td>2D</td>
<td>2D</td>
<td>2.25</td>
<td>1.25</td>
</tr>
<tr>
<td>2</td>
<td>0.139</td>
<td>1210</td>
<td>146</td>
<td>2D</td>
<td>2D</td>
<td>2.25</td>
<td>1.25</td>
</tr>
<tr>
<td>3</td>
<td>0.130</td>
<td>1250</td>
<td>146</td>
<td>2D</td>
<td>2D</td>
<td>2.25</td>
<td>1.25</td>
</tr>
<tr>
<td>4</td>
<td>0.100</td>
<td>1460</td>
<td>147</td>
<td>2D</td>
<td>2D</td>
<td>2.25</td>
<td>1.25</td>
</tr>
<tr>
<td>5</td>
<td>0.100</td>
<td>1350</td>
<td>102</td>
<td>2D</td>
<td>2D</td>
<td>2.25</td>
<td>1.25</td>
</tr>
<tr>
<td>6</td>
<td>0.137</td>
<td>1420</td>
<td>107</td>
<td>2D</td>
<td>2D</td>
<td>2.25</td>
<td>1.25</td>
</tr>
<tr>
<td>7</td>
<td>0.140</td>
<td>1100</td>
<td>89</td>
<td>2D</td>
<td>2D</td>
<td>2.25</td>
<td>1.25</td>
</tr>
<tr>
<td>8</td>
<td>0.075</td>
<td>1500</td>
<td>103</td>
<td>2D</td>
<td>2D</td>
<td>2.25</td>
<td>1.25</td>
</tr>
<tr>
<td>9</td>
<td>0.138</td>
<td>1260</td>
<td>96</td>
<td>2D</td>
<td>2D</td>
<td>2.00</td>
<td>1.00</td>
</tr>
<tr>
<td>10</td>
<td>0.101</td>
<td>1310</td>
<td>105</td>
<td>2D</td>
<td>2D</td>
<td>2.00</td>
<td>2.00</td>
</tr>
<tr>
<td>11</td>
<td>0.137</td>
<td>1290</td>
<td>106</td>
<td>2D</td>
<td>2D</td>
<td>2.00</td>
<td>0.30</td>
</tr>
<tr>
<td>12</td>
<td>0.101</td>
<td>1480</td>
<td>105</td>
<td>2D</td>
<td>2D</td>
<td>2.10</td>
<td>1.10</td>
</tr>
<tr>
<td>13</td>
<td>0.102</td>
<td>1480</td>
<td>104</td>
<td>2D</td>
<td>3D*</td>
<td>2.10</td>
<td>1.10</td>
</tr>
<tr>
<td>14</td>
<td>0.101</td>
<td>1200</td>
<td>108</td>
<td>2D</td>
<td>2D</td>
<td>2.00</td>
<td>1.00</td>
</tr>
<tr>
<td>15</td>
<td>0.138</td>
<td>1370</td>
<td>101</td>
<td>Pipe</td>
<td>2D</td>
<td>2.00</td>
<td>1.00</td>
</tr>
<tr>
<td>16</td>
<td>0.141</td>
<td>1130</td>
<td>103</td>
<td>Pipe</td>
<td>2D</td>
<td>2.00</td>
<td>1.00</td>
</tr>
<tr>
<td>17</td>
<td>0.142</td>
<td>1290</td>
<td>104</td>
<td>Spray P.</td>
<td>Round</td>
<td>2.00</td>
<td>1.00</td>
</tr>
<tr>
<td>18</td>
<td>0.140</td>
<td>1280</td>
<td>111</td>
<td>Spray P.</td>
<td>Round</td>
<td>2.00</td>
<td>1.00</td>
</tr>
<tr>
<td>19</td>
<td>0.100</td>
<td>1180</td>
<td>100</td>
<td>Spray P.</td>
<td>Round</td>
<td>2.00</td>
<td>1.00</td>
</tr>
</tbody>
</table>

*Table 4.3 Overview test conditions*
The complete test program is shown in Table 4.3. During test 1 - 12 both the inflow as well as the overflow were located over the total width of the hopper to create a two-dimensional flow pattern. The overflow used during test 13 had only a width of 1 m instead of the full width of the flume.

4.6 Instrumentation

The instrumentation can be divided into the general (bulk) measurement of the inflow and outflow fluxes and the instruments placed inside the hopper. The following bulk measurements were carried out (the measurement principle between brackets):

- Discharge into the hopper (Electromagnetic Flow meter).
- Inflow concentration (Radioactive Density meter).
- Outflow concentration (Radioactive Density meter).
- Concentration in Mixture Tank (electric conductivity).
- Concentration in U-bend on the suction carriage (Pressure difference).
- Water temperature (electric conductivity).
- Samples of inflow and outflow mixture to determine both concentration as grain size distribution.

At regular intervals samples were taken from the inflow and outflow mixture. From these samples the mixture density and the particle size distribution were determined. The mixture density was used to check and calibrate the radioactive concentration meters.

Inside the hopper the following quantities were measured:

Concentration profiles

On six locations vertical bars were placed each containing 12 gauges for concentration measurement using the electric conductivity principle. This resulted in 72 measurement locations from which 42 locations were sampled during an experiment. The locations of the bars were not varied during the test programme. From the measurements a three dimensional impression of the development of the concentration during time in the hopper could be achieved.

Mixture velocity

During the tests the mixture velocity was measured using 15 Electromagnetic Velocity Meters (E-40 probe developed by Delft Hydraulics). This instrument measures the velocity in two dimensions. Most instruments were placed horizontally, hence velocity was measured in the upward (z) and longitudinal (x) direction, others measured the velocity in a horizontal plane (x, y) (see Figure 4.1 and 4.2 for the definition of the coordinate system). This resulted in 30 velocity signals during a test. Except for one, all gauges were fixed on a certain horizontal (x, y) position before commencement of an
experiment. The vertical position of the gauges could (and must) be varied during a test. During the experiments the hopper was filled with sand. Burying of the instruments should be avoided for two reasons. First of all mixture velocities are only measured above the sand surface and secondly it was very likely that the gauges would be damaged during sand removal from the hopper. When the sandbed reached a certain probe level during filling the instrument was moved in vertical direction over a distance of 0.2 to 0.4 m. The time of relocation and the new position was recorded for later processing of the velocity signals.

4.6.1 Calibration and accuracy

The density meters used to measure the inflow and overflow concentration were calibrated using samples. The electromagnetic flow meter and the electromagnetic velocity meters (EMS) were calibrated separately by WL|Delft Hydraulics using standard procedures.

Special attention was needed for the calibration of the electric conductivity probes used in the hopper. The measurement principle of these instruments is that water conducts electricity in contrast to the suspended sediment. The conductivity of the water phase depends on temperature and dissolved ions, which both vary from test to test. It is therefore not possible to use a unique relation between conductivity and concentration for all tests. To overcome this problem the probes were calibrated every test. Before an experiment was commenced the hopper was pumped full with water and the "clear water" conductivity was measured with every probe. Thereafter the water level in the hopper was lowered to the desired start level. At the end of a test the hopper was full with sand, so all probes were buried. It was assumed that all probes measured the same bed concentration and this value was used for calibration.

4.6.2 Data acquisition

The signals of the instruments were sampled using two PC’s. The data of the bulk-measurements (in- and overflow quantities) were sampled with a frequency of 2 Hz, the signals of the concentration and velocity probes in the hopper were sampled at 10 Hz to preserve information of the turbulence. After a test was finished the first stage of post processing was carried out to convert the signals to physical values and to convert all sampled velocity signals \( u(t) \) to \( u(x, y, z, t) \). The rough signals were written to an ASCII file during the test. Post processing of the data was performed using MATLAB.
4.7 General Observations

4.7.1 Introduction
In this section the results obtained from the experiments are analysed. The information obtained can be split into quantitative information (the test data) and qualitative information based on visual inspection of the process combined with a detailed analysis of the bulk measurements and measurements inside the hopper. For detailed information reference is made to Van Rhee, (1999a). In this thesis a general description of the processes observed inside the hopper will be given in Section 4.7.2. Thereafter the attention is focussed on the quantitative results.

4.7.2 Phenomenological description of the process
The velocity and concentration measurements inside the hopper and visual observation through the glass side wall provided an excellent insight into the loading process. This process can be divided into three stages (Figure 4.5)
1. Filling of the hopper with mixture until the overflow level is reached.
2. The overflow phase.
3. The final phase.
These three stages are described below.

First Phase: Filling to overflow level
When loading starts on board of a TSHD, the hopper is generally partly filled with water, with the water level inside the hopper equal to outside. At the start of most laboratory tests the model hopper was therefore partly filled with water as well (\(h_{\text{start}}\) in Table 4.3). During phase one the level of the mixture in the hopper rises until the overflow level is exceeded and overflowing starts. The flow pattern inside the hopper is strongly influenced by density currents. The density of the mixture discharged in the hopper is greater than that of the fluid inside the hopper. The mixture flows down towards the bed below the inlet section due to this density difference. In the inflow section flow velocity and the level of turbulence is high near the bed. As a result sedimentation in this section is lower compared with downstream (in direction of overflow) locations. As a result a scour hole develops at the inflow section after some time. From the scour hole the mixture flows to the other side of the hopper as a density current. This implies that the flow is concentrated near the bed.

The flow velocity in the density current is relatively high as can be demonstrated with the following example. For a typical test the discharge is 0.1 m\(^3\)/s. At the start of the test the water depth is about 1 m. The width of the hopper is 3.08 m. One could expect a horizontal velocity of approximately 0.03 m/s in case flow velocity distribution in the hopper would be uniform. Instead velocities of up to 0.6 m/s were measured in the density current close to the bed.
Second Phase: Overflow Phase

During this phase the level of the mixture in the hopper remained constant because the discharge and height of the overflow were not varied during a test. The discharge of the overflow is equal to the inflow. A density current is present near the bed for the duration of the second phase. Sediment settles from this current and the sand bed rises gradually. For test 1-4 where 140 μm sand was used the grain settling efficiency (equation (2.3)) was relative high and the majority of the particles in the density current settled on the bed.

![Diagram of overflow phases](image)

*Figure 4.5  Overview different phases during loading process.*

The concentration in the mixture above the density current remained consequently very low. For the other tests executed on 100 μm sand the settling efficiency was lower and a proportion of the particles did not settle and remained in suspension. The concentration in the suspension increased during time and likewise the concentration in the overflow. For most tests the concentration (in suspension and overflow) increased until the bed reached the overflow level. An exception to this general observation is the situation at a very low or a very high influx of sand. In these cases the concentration in the suspension and overflow reached a constant value before the end of the test.
The highest velocities in the density current occurred at the start and the end (see third and final phase) of the loading process. At the start the velocity is high because the large difference between the density of the inflowing mixture and the water initially present in the hopper. This density difference (buoyancy) is the most important driving force for the density current. During a test this driving force diminishes because the density difference diminishes. Another effect is the decreasing distance between the mixture surface and the sand bed with time. Less time is available for the density current to accelerate towards the bed leading to a lower velocity.

Near the mixture surface a two-layer system consisting of a lower above a higher concentrated layer develops after some time. The lower concentrated top layer flows considerably faster towards the overflow than the higher concentrated layer. Due to the large concentration gradient near the surface the direction of the flow is nearly horizontal (see Figure 4.6).

**Third and final Phase**

In the final phase the top of the density current reaches the water surface. Soon after that moment a free water flow (like a river flow) develops and the flow velocity increases considerably. The increased velocity reduces sedimentation and the outflow concentration increases strongly. If loading is continued the outflow concentration becomes equal to the inflow concentration, and the total influx of sediment disappears in the overflow.

### 4.7.3 Flow pattern in the hopper

Figure 4.6 shows the flow pattern in the hopper during the overflow phase in a schematised way. The inflow is located at the left side and the overflow at the right side of the hopper.

The hopper area can be divided into 5 different sections:

1. Inflow section
2. Settled sand or stationary sand bed
3. Density current over settled bed
4. Horizontal flow at surface towards the overflow
5. Suspension in remaining area
Figure 4.6  Schematic overview of flow field in a hopper.

In the inflow section the mixture flows towards the bed and a scour hole is created. A density current transports and distributes the sediment over the length of the hopper. From the density current sediment settles which leads to a rising bed. The vertical velocity of the interface between the bed and the density current is defined as the sedimentation velocity. The sedimentation velocity depends on the concentration in the mixture close to the bed, the settling velocity and a possible reduction due to the bed shear stress from the flowing mixture. The part of the incoming sediment which does not settle moves upward into suspension. At the water surface a relative strong horizontal flow of low concentrated mixture towards the overflow is present. Apart from the inflow section the flow in the suspended part of the hopper is basically one-dimensional in vertical direction. In horizontal direction the (measured) concentration is very uniform (layered).

4.8 Inflow and Overflow Measurements

4.8.1 Introduction

In total 19 tests were executed (see Table 4.3). The attention here will be focussed on test 05, 06 and 08. Test 06 was chosen because during this test the sand influx was largest. Test 05 and 08 represent an intermediate sand influx, but with different discharge-concentration ratios. Details on the other tests are given in Van Rhee, (1999a)

4.8.2 Inflow and overflow concentration

The registration of the in- and outflowing quantities is discussed first. A typical registration of the inflow and outflow concentration can be observed in Figure 4.7. These properties are plotted against loading time. The initial water level in the hopper was situated at 1 m below overflow level, so overflow started only after approximately 500 seconds which was the time needed to reach the overflow level. The inflow
concentration is kept constant as much as possible, though some variation is present due to the suction process from the sand storage basin. The solid lines show the registrations of the density meters. The symbols indicate the density of the samples taken from the inflow and overflow.

![Graph showing concentration vs time](image)

*Figure 4.7 Measured in- and over-flow concentration of test 08.*

The overflow concentration shows a trend observed during most tests: a steadily increasing concentration with time. This has a direct relation with the increasing concentration in the suspension in the hopper as will be shown in Section 4.9.1.

### 4.8.3 Overflow Losses

A very important parameter during the loading process is the overflow loss. Two different definitions of this parameter are used. The loss can be defined as the ratio of the outflow and inflow sediment flux at a certain moment, or as the ratio of the total outflow and inflow sediment volume.

The overflow flux is defined as:

$$OV_{flux} = \frac{c_o(t)Q_o(t)}{c_i(t)Q_i(t)}$$  \hspace{1cm} (4.24)

In which $c$ is the volumetric concentration and $Q$ the discharge. The indices $o$ and $i$ relate to the overflow and inflow respectively. The (cumulative) overflow loss is defined as:

$$OV_{cum} = \frac{\int_0^r c_o(t)Q_o(t)\,dt}{\int_0^r c_i(t)Q_i(t)\,dt}$$  \hspace{1cm} (4.25)

Figure 4.8 shows the measured overflow flux and cumulative overflow loss as function of time. The observed behaviour was found to be typical for most of the experiments.
The cumulative overflow losses and the overflow flux both increased with time. The gradually increasing overflow flux with time indicates that the loading time was not sufficient to achieve a stationary situation. Just before the hopper is fully loaded a dip in the overflow flux can be observed. (In Figure 4.8 at t=2000 s). This is the moment that the top of the density current reached the water surface at which the two-layer system at the surface disappeared. As a result a large amount of low concentrated water was pushed out of the hopper.

![Overflow flux and cumulative](image)

**Figure 4.8**  Overflow flux and overflow loss versus time (test 08).

### 4.8.4 Overflow rate

In the previous Section the definition of the overflow loss was presented. It is now investigated which parameters influence these losses. In Section 2.3 the grain settling efficiency was defined (Camp, 1946).

\[
r_g = \frac{w_i}{v_0}
\]  

(4.26)

The overflow rate (or hopper load parameter) is defined as:

\[
v_0 = \frac{Q}{BL}
\]  

(4.27)

Where \( Q \) is the discharge and \( BL \) the surface area of the hopper. The overflow rate can be made dimensionless using the settling velocity of the sediment. When the influence of hindered settling is included by using the relation of Richardson and Zaki, (1954), the dimensionless hopper load parameter \( H^* \) can be written as (Wijant, 1993):

\[
H^* = \frac{v_0}{w_i} = \frac{Q}{w_0 (1-c)^n BL}
\]  

(4.28)

In this equation \( w_0 \) is the settling velocity of a single grain, \( c \) the volume concentration and the exponent \( n \) depend on the particle Reynolds number. (see Section 3.4). The

70
grain settling efficiency is defined for a certain grain diameter within a particle size distribution. The settling velocity used in the definition of $H^*$ is based on the $D_{50}$ of the particle size distribution. Both $H^*$ and $r_g$ represent a ratio between two velocities.

### 4.8.5 Dimensionless overflow rate $S^*$

Instead of comparing velocities, a hopper load parameter can be derived on the basis of sand fluxes. In that case the sandflux in the inflow can be compared with the sandflux settling on the bed. The inflowing mass flux equals:

$$s_m = \rho_s Q c_{in} \quad (4.29)$$

Where $c_{in}$ is the inflow concentration. The massflux settling on the bed becomes:

$$s_{sed} = \rho_s \left(1 - n_0\right) BL v_{sed} \quad (4.30)$$

In this equation $n_0$ is the porosity and $\rho_s$ the density of the grains. The sedimentation velocity $v_{sed}$ (vertical velocity of bed) was defined in Section 3.11 as:

$$v_{sed} = \frac{S - E}{(1 - n_0 - c_b) \rho_s} \quad (4.31)$$

Where $n_0$ is the porosity of the bed, $c_b$ the near bed concentration and $S$ the settling flux:

$$S = \rho_s c_b w_z \quad (4.32)$$

The pickup rate $E$ is usually made dimensionless by:

$$\phi = \frac{E}{\rho_s \sqrt{g \Delta D}} \quad (4.33)$$

The dimensionless overflow rate $S^*$ is defined as the ratio between the inflowing and settling mass flux. Combining equations (4.29) - (4.33) yields:

$$S^* = \frac{s_{in}}{s_{sed}} = \frac{1 - n_0 - c_b}{1 - n_0} \frac{Q}{BL} \frac{c_{in}}{c_b \left(c_b w_z - \phi \sqrt{g \Delta D}\right)} \quad (4.34)$$

In case erosion can be neglected, the expression for $S^*$ can be simplified to $S^*$ (Van Rhee, 2001a): :

$$S^* = \frac{c_{in} \frac{1 - n_0 - c_b}{c_b}}{1 - n_0} \frac{Q}{BL w_z} = \frac{c_{in} \frac{1 - n_0 - c_b}{c_b}}{1 - n_0} H^* \quad (4.35)$$

In the above expression the total concentration (sum of the fractional concentrations) is used for $c_{in}$ and $c_b$. The settling velocity is based on $D_{50}$ of the particle size distribution.
4.8.6 Relationship between cumulative overflow loss and overflow rate

A relationship between the overflow losses and the dimensionless overflow rate is expected with the measured cumulative overflow losses at the end of a test (fully loaded hopper). However the near bed concentration $c_b$ is not known a priori. As a first approximation $c_b = c_m$ can be used. The cumulative overflow loss is plotted versus $H^*$ and $S^*$ (so without the influence of erosion) in Figure 4.9 for all experiments carried out in the present investigation. This figure shows clearly that, as can be expected, the losses increase with the hopper load parameter $H^*$ though a lot of scatter is present and the results of the 140 and 105 $\mu$m sand do not fit entirely. When the right panel is observed it is clear that a much better relationship between losses and dimensionless overflow rate is achieved when $S^*$ is used.

![Graph showing cumulative overflow loss versus dimensionless overflow rate](image)

*Figure 4.9  Cumulative overflow loss versus dimensionless overflow rate.*

The best fit through the data points can be expressed as:

$$OV_{\text{cum}} = 0.39(S^* - 0.43)$$  

(4.36)

The following influences are not included in the value of $S^*$:
- The loading and overflow structure.
- The horizontal transport of sediment (density current).
- The shape of the grain size distribution.

Nevertheless the overflow loss in the model is predicted quite well by equation (4.36). This indicates that these influences did not play a major role during the experiments. It is not clear however if this applies for prototype situations as well. It will be shown later that some of the influences can affect overflow losses. It is therefore not clear if the simple equation above can be applied in general, or is restricted to model scale. It should be regarded as a simple method to achieve a lower limit for the overflow losses.
4.8.7 Particle size distribution

As mentioned in Section 4.6 samples were taken from the inflow and overflow. From these samples Particle Size Distributions (PSD) were determined using the laser diffraction principle. The results from test 08 are shown in Figure 4.10. The PSD of the inflow was reasonably constant but the overflow samples showed a large variation of the PSD. The sequence of the sampling in the overflow is indicated in the figure and it is clear that the sand in the overflow becomes coarser during time. At the end of the test, the difference between the PSD's of the overflow and inflow samples diminish. The increasing grain diameter in the overflow is related to the increasing concentration in the overflow. Due to hindered settlement the settling velocity decreases with concentration and therefore larger grains remain in suspension and are removed with the overflow.

![Graph showing PSD in and PSD out with time intervals](image)

*Figure 4.10  Particle size distributions from inflow and overflow samples from test 08.*

4.9 Measurements inside the hopper

4.9.1 Development of Concentration Profile during time

Using the electric concentration gauges inside the hopper the development of the concentration during time can be monitored. In Figure 4.11 concentration profiles in the middle of the hopper (x = 6 m) are plotted at a time interval of 120 s. The concentration was measured at seven different vertical positions (per x, y location), hence vertical resolution is limited. Near the sand bed a large concentration gradient is present which cannot be measured accurately due to this limited resolution. The resolution near the bed can however be improved using the following method.
During a relative short period the concentration distribution can be regarded constant relative to the vertically moving sand surface. The sedimentation velocity can be determined from the moments that the gauges are buried in the sand bed. The concentration signal as function of time can therefore be transformed as function of vertical distance using the sedimentation velocity. This transformation is only used to estimate the concentration distribution between two gauges where the sand surface is present. For the remaining part of the profile the directly measured data is used. The resulting profiles are shown in Figure 4.11.

![Figure 4.11 Concentration profiles test 05 at 120 s interval (x=6 m).](image)

Every concentration profile is composed of the direct measured and the transformed part. The part of the profile obtained through this transformation is marked with “o” in Figure 4.11. The interface between the settled sand and the suspension can be seen very clearly in Figure 4.11. The increasing suspension concentration as a function of time is very evident as well. The same Figure shows that all probes measure a settled bed concentration of 0.55 (porosity = 0.45). This is not coincidental since this value is prescribed and used for calibration of the probes (see Section 4.6.1).

### 4.9.2 Spatial Distribution of the concentration

In the previous Section the development of the concentration during time at one horizontal location \((x, y)\) was shown. The spatial distribution is also important. Figure 4.12 shows concentration profiles at three different horizontal positions \((x = 3, 6, 9\) m) for three different points in time. From this Figure the following observations can be made:

- During loading very little variation of the concentration in a horizontal plane is present in the suspension.
For this test the bed slope is nearly horizontal between $x = 3$ m and $x = 6$ m, between $x = 6$ m and $x = 9$ m the bed had a gentle slope. The bed slope varied from test to test, but all experiments showed that concentration gradients in a horizontal plane are very small, except close to the settled sand bed where a density current is present.

**Figure 4.12** Concentration profiles at different horizontal positions (test 05).

### 4.9.3 Sedimentation velocity

The sedimentation velocity can be deduced directly from the concentration measurements in the hopper by the difference in time between the moment of burial of two adjacent probes. In equation (4.31) a relationship for the sedimentation velocity was given. It is now investigated if this relationship agrees with the experiments. When the influence of the pickup rate is omitted the equation reads:

$$v_{sed} = \frac{w_0 c_b}{1 - n_0 - c_b} = \frac{w_0 c_b (1 - c_b)^n}{1 - n_0 - c_b}$$

(4.37)

To compute the sedimentation velocity the near bed concentration $c_b$, the settling velocity $w_0$, the porosity $n_0$ and the exponent $n$ of the hindered settling function must be known. Different formulas can be used to determine $w_0$ and $n$, as was shown in Chapter 3. The near bed concentration $c_b$ was measured, but the calibration of the probes was based on a prescribed value of the porosity. For all tests $n_0 = 0.45$ was used. The measured sedimentation velocity is plotted versus the near bed concentration in Figure 4.13. Beside the measurements the theoretical relationship (4.37) is shown for three different values of the porosity of the settled bed. The settling velocity $w_0$ was based on equation (3.4), and the exponent $n$ of the hindered settling function according to the original value proposed by Richardson & Zaki (1954).
The measurements are situated below the theoretical line for $n_0 = 0.45$, and match $n_0 = 0.40$ better. Such a low value for the porosity is however not realistic, since this type of sand (fine and uniform) settles in a very loose state. In Figure 4.14 the same data is compared with the theoretical sedimentation velocity using the hindered settling exponent according to equation (3.16) (Garside et. al., 1977). The exponent according to Garside et. al. is somewhat higher compared with Richardson and Zaki. This implies a stronger hindered settling effect. Accordingly the data fits the relation for $n_0 = 0.45$ quite well in Figure 4.14.

**Figure 4.13** Sedimentation velocity versus near bed concentration (test 08), hindered settling exponent according to Richardson and Zaki.

**Figure 4.14** Sedimentation velocity versus near bed concentration (test 08), hindered settling exponent according to Garside et. al.
Summarising it can be concluded that the measured sedimentation velocity agrees well (using reasonable assumption regarding the porosity, settling velocity and hindered settling exponent) with the theoretical relation of equation (4.37). This equation does not include the influence of erosion. This gives another indication that erosion did not play an important role during test 08. The results of the other tests show the same tendency, which implies that this indication is valid for the complete test programme.

4.9.4 Flow velocity inside the hopper

The flow velocity in the hopper was measured in two directions at 15 different \((x,y)\)-positions. At the beginning of an experiment the instruments were located at a vertical start position. When the sandbed reached this position the probe was lifted over a distance of 20 cm. A typical registration of the horizontal velocity is shown in Figure 4.15 (lower panel). The upper panel of this figure shows the vertical position of the probe versus time.

\[\text{Figure 4.15} \quad \text{Horizontal velocity measured by EMS 1 during test 08 (x=3 m).}\]

At the start of the test the instrument was located close to the bottom at \(z = 0.3\) m. During the experiment the EMS was relocated nine times (different line segments). After repositioning the EMS records a value for the horizontal velocity which drops to zero after some time when the instrument is buried in the sand. The figure shows clearly that the maximum measured velocity gradually decreases during time in spite of the fact that at the same time the flow depth (distance between bed and surface) decreases. This can be explained by the fact that the flow is driven by density gradients. During time the concentration in the suspension increases. The density of the inflow mixture remains approximately constant during the test implying that the driving force for the density current diminishes.

77
Figure 4.16  Horizontal velocity measured by EMS 2 during test 08 (x=3 m).

It is interesting to note that in most hopper sedimentation models the opposite trend of the velocity near the bed is assumed. Due to the assumed uniform or logarithmic velocity distribution in these models, the reduction of the flow depth with time leads to increasing velocity (near the bed) with time. Another registration from the same test is shown in Figure 4.16. This instrument was initially located at a larger distance from the bottom (z = 1.25 m) at the start of the test. From the start of the test up to approximately t = 700, a small negative flow velocity (towards the inflow side) is recorded. The negative flow velocity at a larger distance above the bed implies that the flow circulates in the hopper. The negative velocity was also present in the hopper after t = 700, but not recorded with EMS 2, because the instrument was not relocated high enough above the bed after that time.

4.9.5  Velocity profiles as function of depth

The same method used to convert the time registration to a concentration profile (Section 4.9.2) can be applied to the velocity signals. Assuming a stationary flow and sedimentation velocity during a limited time, the measured flow velocity as function of time can be transferred to a function of vertical distance. Horizontal flow velocity profiles were computed and are shown in Figure 4.17. In this figure four profiles are plotted at different locations.

It is clear that only at x = 8 m the registration time was long enough to measure over the complete depth of the density current. For the other locations only the lower part of the profile could be constructed. The thickness of the density current is approx. 0.25 m and the maximum velocity is situated close to the bed. Since 19 tests were performed, and the conversion from time to space can be made at any given time it is obvious that
an almost unlimited number of profiles can be plotted. Here only one is shown to illustrate the principle and to show the order of magnitude of the velocities measured. In Chapter 7 where the 2DV model will be compared with the experiments more results are shown.

![Graph showing velocity profiles during test 05.](image)

**Figure 4.17** Velocity profiles during test 05.

### 4.10 Influence of inflow and overflow structures

To analyse the influence of the inflow and overflow configuration, the cumulative overflow loss is plotted as a function of the inflowing sandflux in Figure 4.18.

![Graph showing cumulative overflow loss for 100 μm sand as function of sandflux.](image)

**Figure 4.18** Cumulative overflow loss for 100 μm sand as function of sandflux.
The inflow and overflow configuration was only varied for 100 \( \mu \)m sand, so only these test results were used. The losses are plotted for the tests with a discharge of approximately 0.10 m\(^3\)/s and 0.14 m\(^3\)/s. For a certain discharge (symbol) the other variations were the inflow concentration and in- and overflow configuration. It is assumed that for constant inflow and outflow configuration and constant discharge a smooth relation between the overflow loss and the sandflux exists since the only independent parameter in that case is concentration. The influence of these configurations should therefore be visible from the recorded overflow losses. When we for instance concentrate on all tests executed with a discharge of 0.14 m\(^3\)/s, we see that all data points are on one line except for the result of test 11. The overflow loss for this experiment is less than the over-all relationship, hence this is a favourable situation. The difference between test 11 with all the other tests is the initial height of the water level in the hopper (see Table 4.3). The observation that all other data points are on one line leads to the conclusion that the influence of the in- and outflow configuration was negligible. We must however be very careful to extend this conclusion to prototype scale because there might be scale effects (erosion not of the right scale, see Section 4.3.1). The result of test 11 indicates that a low initial water level in the hopper is favourable. However only one data point is present to support this conclusion.

In Figure 4.18 it can also be seen that the influence of the discharge on the overflow loss diminishes when sandflux increases.

### 4.11 Concluding Remarks

In this Chapter, the large-scale hopper sedimentation tests have been analysed. These tests have improved insight in the processes inside the hopper. Density currents dominate the flow pattern in the hopper. This phenomenon was already expected in the past, but has now been quantified. Beside the improved insight, the experiments have produced high quality data regarding overflow quantities and concentration and flow velocity in the hopper.
Chapter 5
Closed Flume Experiments

5.1 Introduction

The sedimentation velocity determines the time needed to fill a hopper with sand. In Section 3.11 an expression for the sedimentation velocity was presented. For convenience this relation is shown here again:

\[ v_{sed} = \frac{S - E}{(1 - n_0 - c_b) \rho_s} \]  \hspace{1cm} (5.1)

Where \( S \) is the settling flux, \( E \) the pickup rate, \( n_0 \) the porosity of the bed and \( c_b \) the near bed concentration. When this simple equation is examined we can distinguish the following two extreme situations:

\( S \gg E \) The settling flux is much larger than the pickup rate. This situation occurs at stagnant flow condition, for instance in one-dimensional sedimentation in a sedimentation column without the generation of turbulence. In that case the sedimentation velocity can be computed directly using the near bed concentration and settling velocity (equation (4.37)).

\( E \gg S \) The pickup rate is higher than the settling flux. In that case net erosion takes place.

The second situation is not very likely to occur in a hopper, because normally an overall settling behaviour is present. It is however not unlikely that erosion and settling flux are of the same order of magnitude, implying that the sedimentation is hindered (reduced) by the bed shear stress. In the previous Chapter we have seen that for the model tests this reduction did not play an important role (see Section 4.9.3). In prototype situation the situation might be different. It is therefore important to determine the magnitude of the pickup rate for the circumstances in a hopper of high sediment concentration in combination with a relative low bed shear stress.

Very little can be found on this subject in literature (for the circumstances in a hopper). So it was decided to set up an experimental program. In the previous Section it was argued that the erosion and sedimentation in a model test are not on the same scale. To avoid scale effects, it is important to achieve flow velocities of the same magnitude as present in prototype situation. It is difficult to achieve the same velocity as in prototype when a set-up with a free-surface flow is chosen. This is caused by the fact that the driving force for the flow in the hopper is gravity (density currents). The sandflux per
unit width in the model should be the same as in prototype in order to get the same velocity as in prototype. This would lead to large experimental dimensions and costs. When on the other hand the flow is confined in a closed flume (no free-water surface) the driving force is the pressure gradient enforced by the pump installed in the closed circuit. Therefore a larger flow velocity can be reached with limited dimensions of the experimental set up.

The experiments executed at WL/Delft Hydraulics and Delft University of Technology are described in this Chapter.

5.2 Closed Flume Pilot tests

The basic approach of the test was to pump a concentrated sand-water mixture at a velocity higher than the deposition velocity through a closed circuit. The deposition velocity is the velocity at which a stationary (settled) bed is formed in a pipeline. When the flow velocity is above this value all sediment remains in suspension. The circuit contained a section for measurements with a (transparent) rectangular cross section (height 100 mm, width 50 mm). Parallel to this section a by-pass pipe was mounted. The flow velocity in the test section could be lowered in a relative short time with a butterfly valve. At the same time the velocity increased in the by-pass section.

![Diagram of closed flume experiment](image)

*Figure 5.1 Schematic overview closed flume experiments at WL/Delft Hydraulics.*

The experiments were performed as follows. The sand-water mixture was pumped through the system via the test section and by-pass at a velocity well above the deposition limit. By closing the valve upstream of the test section, the flow velocity in this section was suddenly decreased to a value below the deposition velocity (for some tests the valve was completely closed, so velocity was reduced to zero m/s). Hereafter more sediment was in suspension than would be in an equilibrium situation at that low velocity. Hence the flow was overloaded with sediment and sedimentation occurs.
During the experiments sedimentation could take place at different combinations of mean concentration and bed shear stress. The latter can be varied from zero (valve completely closed) up to a value close to the critical value for the deposition velocity. The experiments simulated for a short time the situation close to the bed in a hopper.

The advantage of the set-up, apart from the modest dimensions, is that a large number of tests can be performed in a short time in contrast to the model hopper sedimentation tests described in the previous Chapter. After a test was executed in the closed flume sediment had settled in the test section. The sediment was brought into suspension again by opening the valve to the test section.

Some disadvantages are also present with this approach. The flow in the rectangular pipe section is different from the density current in a hopper. In the hopper the height of the density current is much smaller than the width, while during the closed flume tests both width and depth of the flow are of the same order of magnitude. This implies that wall influences are more significant during the flume tests. Another difference is the presence of the wall on top of the flow during the closed flume tests. Nevertheless it was decided to perform these tests because they provided quantitative information on the situation close to the bed at acceptable cost. To investigate if this type of arrangement would provide satisfactory results, the principle was tested in a small flume at WLI Delft Hydraulics (van Rhee & Talmon, 2000). Test were performed on sand with a $D_{50}$ of 110 and 200 μm.

### 5.2.1 Measurements

The pilot tests were executed with simple instrumentation. The flow velocity was measured using two Electromagnetic Flow meters. One was used to measure the total discharge in the system, the other measured the discharge in the by-pass pipe. From these measurements the discharge in the test section was obtained. The pressure difference over the test section was not measured. The bed shear stress was therefore not measured directly, but estimated from the flow velocity. The (mean) concentration was measured by taking samples from the flow. The temperature was measured using a thermometer. The sedimentation velocity was measured visually using a video camera. The time needed to form a certain bed height (10 mm) was measured using a stopwatch.

### 5.2.2 Results pilot tests

For a more detailed description of the experiments and results, reference is made to Van Rhee, 1999 and Van Rhee & Talmon, 2000. Here, the most important results are highlighted. The measured sedimentation velocity is plotted versus the flow velocity for 110 μm sediment in Figure 5.2. Lines are plotted for different mean concentration. The sedimentation velocity decreases with increasing flow velocity as was expected.
For the highest concentration (30%) the difference between the settled bed and the mixture above was very difficult to see on the video recordings. The results for this concentration are therefore unreliable. A remarkable phenomenon was observed at low flow velocity. The sedimentation velocity at a flow velocity of approximately 0.3 m/s is somewhat higher compared with the sedimentation velocity at stagnant condition (zero flow velocity).

![Graph showing sedimentation velocity versus flow velocity](image)

**Figure 5.2** Sedimentation velocity versus flow velocity ($D_{50}=110 \mu m$).

Only the test results of the 110 \( \mu \)m sand are shown in this thesis. The sedimentation velocity of 200 \( \mu \)m sand was too high for the pilot test set-up. Sedimentation started already during closing of the valve. Before the lower flow velocity was established in the test section the larger part of the suspended sediment had already formed a bed. It was therefore not possible to find a relation between flow and sedimentation velocity for this sand during these tests.

### 5.2.3 Pickup at high concentration

The measured reduction of the sedimentation velocity \( v_{sed} \) can be expressed as a pickup rate using equation (5.1):

\[
E = S - \rho_z (1 - n_0 - c_b) v_{sed}
\]

(5.2)

Where the settling flux \( S \) follows from the product of the bed concentration and the settling velocity (including hindered settling):

\[
S = \rho_z c_b w_z
\]

(5.3)

The settling flux can be computed when the settling velocity and near bed concentration \( c_b \) is known and the porosity of the settled bed is estimated. The value of \( c_b \) was not measured. Therefore the mean concentration was used. This value was used as well to
compute the settling velocity \( w_s \). The sedimentation velocity is measured so the pickup rate can be determined. It is convenient to make the pickup rate dimensionless as follows:

\[
\phi_p = \frac{E}{\rho_s \sqrt{g \Delta D}}
\]  

(5.4)

In Van Rhee & Talmon, 2000 this analysis was executed and the calculated pickup was compared with three different pickup functions (Van Rijn, 1984, Fernandez-Luque, 1974 and Cao, 1997). In this thesis the calculated pickup rate for the tests on 110 μm sand are shown in Figure 5.3, and compared with the pickup function of Van Rijn (1984). In the figure the dimensionless pickup rate is plotted versus the Shields parameter, which is a dimensionless value for the bed shear stress.

\[
\theta = \frac{u^2}{\Delta g D} = \frac{f}{8} \frac{u^2}{\Delta g D}
\]  

(5.5)

Where \( u_s \) is the friction velocity, \( \Delta \) is the specific density defined as \( (\rho_s - \rho_w)/\rho_w \), \( \bar{u} \) the mean flow velocity and \( f \) the Darcy-Weisbach friction factor. The friction factor was estimated using the expression of White-Colebrook. This expression needs a roughness height \( k \). A value of \( 2 \times D_{50} \) was used for this parameter. It is clear that the pickup function according to Van Rijn overestimates the pickup (or reduced sedimentation due to the bed shear stress) for high concentration. The other pickup function mentioned above showed the same tendency.

![Figure 5.3](image)

Figure 5.3 Calculated pickup at high concentration.

The relative high values of the computed pickup rate according to Van Rijn is most probably caused by the fact that this empirical function is based on experiments with low (almost zero) concentration and with Shields values up to unity. In Figure 5.3
negative values for the back-calculated pickup rate for Shields values below unity are observed. By definition the pickup flux should be positive and apparently one or more of the quantities (settling velocity, concentration or porosity) estimated to compute the pickup rate does not have the right value.

5.2.4 Reduction factor

The net sedimentation rate was determined in the previous section as the difference between the settling and pickup flux. When we define \( S_n \) as the net sedimentation flux \( (S_n = S - E) \), the sedimentation velocity reads:

\[
\frac{v_{\text{sed}}}{\rho_s (1 - n_0 - c_0)} = \frac{S_n}{S_0}
\]  

(5.6)

When the sedimentation flux in stagnant flow condition (no erosion) is defined as \( S_0 \), we can write:

\[
\frac{v_{\text{sed}}}{v_{\text{sed},0}} = \frac{S_n}{S_0} = R
\]  

(5.7)

Where \( v_{\text{sed}} \) is the measured sedimentation velocity at mean velocity \( \bar{u} \), \( v_{\text{sed},0} \) the computed sedimentation velocity at zero flow velocity and \( R \) a reduction factor. When a relationship between the reduction factor and shear stress and concentration can be found \( (R = R(\theta, c_0)) \), the sedimentation velocity can be determined by the product of the reduction factor and the sedimentation velocity at zero flow velocity.

![Graph](image)

**Figure 5.4** Reduction of sedimentation velocity versus (dimensionless) shear stress.

This approach can only be applied in flow conditions where net sedimentation occurs, and may provide better results than produced with the method described in the previous
section. In Figure 5.4 the computed reduction (equation (5.7)) is plotted as a function of the Shields parameter $\theta$. The figure shows a more or less linear decrease of the reduction factor with the bed shear stress. In Figure 5.4 it can be seen that for $\theta = 0$ the value of the reduction factor is not unity, as one would expect. This is caused by a difference between the calculated and measured sedimentation velocity at zero flow velocity. For the reduction factor the results from the 30% concentration tests seem to be inconsistent with the other results. This is probably caused by the fact that it was very hard to measure the level of the sand bed from the video recordings at high concentration.

The following mechanisms can be responsible for the reduction of the sedimentation velocity:
- Turbulence. The level of turbulence increases with flow velocity. The turbulent eddies eject the solid particles into the flow.
- Grain-grain interactions. Figure 5.4 shows that the value of the Shields parameter reaches values up to 4.5. In that case a thin layer with high sand concentration is moving in a sheet along the bed (Sheet flow). In that case inter-granular forces can be responsible for the reduction of the sedimentation velocity.

5.3 Closed flume sedimentation tests at TU Delft

The results from the pilot sedimentation tests at Delft Hydraulics were encouraging, so the existing slurry circuit of the Dredging Technology Research Laboratory of TU Delft was modified to execute the same type of experiments (Teheux, 2000). Compared with the pilot tests the following differences can be mentioned.
- The discharge is larger enabling a measurement section with a larger cross section.
- More instrumentation was installed. In the measurement section an EMS was installed to measure the velocity and turbulent fluctuations inside the flume. In total 56 conductivity probes were placed in the wall of the flume to measure the location of the sand bed and the concentration in the suspension above the bed. The pressure difference over the flume was measured from which the shear stress on the bed can be computed. A vertical adjustable radioactive concentration gauge was installed to measure the concentration profile to calibrate the conductivity probes.
- All pipes (measurement section, by-pass and return section were mounted in one horizontal plane to avoid blockage in vertical sections. This problem occurred at the pilot test arrangement.

5.3.1 Layout of test arrangement.

The existing slurry circuit was extended with an arrangement as shown in Figure 5.5. The upper part of the figure shows a top view, where the measurement section and the shunt pipeline can be seen.
Figure 5.5  Top and side view of test arrangement.
The bottom panel shows the side view. The internal dimensions of the rectangular measurement section were 88 × 288 mm and it was connected with a diffuser to the 150 mm pipe circuit. In the centre of the rectangular section a 1500 mm long perspex section was mounted. However during the first tests it became clear that this transparent section could not resist the pressure waves resulting from opening or closing of the butterfly valve. It was therefore decided to replace this section with a steel pipe containing a transparent (lexan) window on one side. This window enabled visual inspection of the flow, and was also required as a non-conducting wall for the concentration probes (see Figure 5.6).

Figure 5.6  Window in measurement section.

5.3.2 Instrumentation

During the test the following quantities were measured:
- Discharge through the measurement section using an EMF.
- Flow velocity at one location in the measurement section using an EMS.
- Mean concentration in the vertical pipe section using a radioactive concentration meter.
- Concentration in the measurement section to determine the concentration profile in stationary condition using a radioactive concentration meter that was adjustable in vertical direction. The information from this instrument was used to calibrate the 2-point concentration conductivity probes.
- Differential pressure gauge to determine the hydraulic gradient over the measurement section.

For optimal resolution the vertical distance between the probes is only 5 mm. To avoid interference between different probes they are positioned in the way of a staircase as indicated in Figure 5.7.
5.3.3 Particle size distributions

Four different sands were used for the experiments with median grain diameter of 125, 150, 185 and 270 μm. The particle size distributions are plotted in Figure 5.8. The continuous lines in this figure are the average graphs for all samples of that specific type of sand.

5.3.4 Results of the test on 125 μm sand.

A typical registration of the measured concentration during a test is shown in Figure 5.9. In this Figure the measured concentration of ten conductivity probes is plotted together with the measured concentration using the radioactive concentration meters.
The concentrations measured with the conductivity probes are plotted with the solid lines. The dotted lines show the concentration measured with the radioactive devices.

![Figure 5.9](image-url)  
*Figure 5.9  Concentration versus time at various height in test flume (D_{50}=125 \mu m, c=25 \%) .*

The conductivity probes are placed at different vertical positions. Sedimentation takes place after the velocity is decreased in the test section. The registered value of every probe starts at 0.25, which is the suspension concentration. After some time a quick transition to a value of 0.5 marks the moment that the sand bed passes the probe. The radioactive concentration device mounted in a vertical pipe section shows a constant concentration of approximately 0.25. The radioactive concentration device is placed in the measurement section close to the bottom and is used to measure the value of the bed concentration. This value is used to calibrate the conductivity probes.

Since the vertical position of every probe is known, the time difference between the transition of subsequent probes can be used to compute the vertical velocity of the sand bed (=sedimentation velocity). The result of this procedure is plotted in Figure 5.10. The left panel shows the computed sedimentation velocity using probes 1 and 3 located on 5 mm and 15 mm above the flume bottom. The right panel is based on probes 2 and 4 located on 10 and 20 mm above the bottom of the flume. The differences between the two figures are small. Overall the sedimentation velocities in the left panel are somewhat smaller, because these measurements are still influenced by the transition of the flow (to a lower velocity). Hence the situation during closing is influencing the results. To avoid the transition phase only the results from probe 2 and 4 will be used in the remainder of this Chapter. For every concentration two lines are drawn (indicated as "high" and "low"). This annotation relates to the value of the flow velocity in the flume.
before the valve is closed (respectively 3.7 m/s and 2.7 m/s for the tests on 125 μm sand). The measured sedimentation velocity for a high initial flow velocity is somewhat smaller. The higher level of the turbulence still present in the flow short after closing the valve is responsible for this difference.

![Graph showing sedimentation velocity versus flow velocity for different concentrations and probe locations.](image)

**Figure 5.10**  Sedimentation velocity versus flow velocity (125 μm).

The sedimentation velocity decreases with increasing flow velocity as was the case for the pilot tests. At a certain velocity (for this sand and pipe geometry of the order of 1.25 - 1.5 m/s) sedimentation velocity was equal to zero. This value is called the critical velocity (or deposition velocity) in pipeline transport literature. Between 0 - 0.5 m/s the influence of the flow velocity on the reduction of the sedimentation velocity is low.

The flow velocity measured in the density current at the model hopper sedimentation tests analysed in Chapter 4 was lower than 0.3 – 0.5 m/s most of the time. Therefore erosion did most probably not play an important role during these test. This was already argued in Chapter 4. For this regime the sedimentation velocity at low concentration shows a different behaviour compared with high concentration. The influence of the reduction due to the bed shear stress is counter balanced by the increased influx of sediment in the measurement section (Van Rhee, 2002).

**5.3.5 Relation between shear stress and flow velocity**

The pressure difference over the measurement section was recorded. This value is used to compute the wall shear stress. For stationary flow, the relation between wall shear stress and pressure gradient reads:

\[
A \frac{dp}{dx} = \tau_w \cdot O
\]

(5.8)
Where $A$ is the area of the pipe cross section and $O$ the wetted perimeter. The index "Δ$\rho$" is added to indicate that the shear stress is calculated from the pressure difference. The relationship between the shear stress and flow velocity for clear water as measured is plotted in Figure 5.11. In the Figure data points are present for different initial (before closing the valve) values for the velocity in the measurement section. The operating procedure was as follows. The valve to the measurement section was fully opened and pump revolutions were chosen to get a velocity between 1.6 and 3.6 m/s. Subsequently the valve was closed until a desired velocity in the measurement section was achieved.

![Figure 5.11](image)

**Figure 5.11** Wall shear stress for clear water.

Hence a certain flow velocity (horizontal axis in Figure 5.11) was obtained with different openings of the valve, depending on the velocity $v_o$ for full open valve. This procedure was followed to investigate the influence of the valve position on the measured pressure difference. As can be seen from the Figure it appeared that in the region between 1.5 – 3 m/s different values of the shear stress at the same velocity were obtained. In that case the valve opening influenced the pressure difference. Fortunately, most tests with sand were carried out at lower velocities so this phenomenon does not influence test results with sand. In the same Figure two lines are plotted with a theoretical relation between shear stress and flow velocity using the relation:

$$\tau_{\Delta \rho} = \frac{1}{2} \rho u^2 \frac{f A}{D O}$$  \hspace{1cm} (5.9)

Where $D$ is the pipe diameter and $f$ the friction factor. The most widely used formula to determine the friction factor for pipe flow is from Colebrook, (1938):
\[ \frac{1}{\sqrt{f}} = -2\log \left( \frac{k/D_h}{3.7} + \frac{2.51}{Re\sqrt{f}} \right) \]  

(5.10)

Where:

- \( D_h \) Hydraulic diameter
- \( k \) Roughness height
- \( Re \) Reynolds number \( Re = uD/\nu \)

Iteration is needed to compute \( f \) from this transcendental equation. Miller (1985) suggested that a single iteration produces a result within 1% if the initial estimate is calculated from:

\[ f_0 = 0.25 \left( \frac{k/D_h}{3.7} + \frac{5.74}{Re^{0.9}} \right)^2 \]  

(5.11)

The hydraulic diameter for a non-circular cross section is defined as:

\[ D_h = \frac{4BH}{2B + 2H} \]  

(5.12)

The factor 4 is introduced so that the hydraulic diameter is equal to the duct diameter for a circular geometry. This leads to a hydraulic diameter of the flume equal to 0.13 m. The roughness height \( k \) for the steel walls is approximately 0.05 mm. Using these values the computed wall shear stress is plotted in Figure 5.11. It is obvious that the measured stresses are somewhat higher and correspond to a lower hydraulic diameter of 0.115 m. The next step is to analyse the relation between shear stress and flow velocity for sand-laden flows. For the tests performed with sand-water mixtures the pressure gradient was measured as well. The shear stress is determined again using equation (5.8). This procedure is not completely correct since the roughness of the bottom and other walls is different due to the presence of a bed. The measured shear stresses for the experiments with 125 \( \mu m \) are compared with the clear water measurements in Figure 5.12. In this Figure the measured shear stress for clear water (data of Figure 5.11) is compared with the measured shear stress during sedimentation (same tests as plotted in Figure 5.10). Note that the stationary situation of clear water flow is compared with a sand-water flow where sedimentation occurs. During the latter tests the flow velocity is constant, but the concentration profile is not in equilibrium. It is clear that the shear stresses are higher than in water, but it is striking that the influence of the concentration is very low. Pressure losses and hence shear stresses increase with the density of the mixture when the flow velocity is above the deposition limit.

Before the valve upstream of the test section is (partly) closed the concentration profile was in equilibrium in the test section. During that time the pressure gradient over the
test section was measured, thus shear stresses in equilibrium mixture flow are also available.

![Graph showing shear stress vs. velocity](image)

**Figure 5.12** Shear stress of sand-water flow velocity during sedimentation compared with water resistance.

These equilibrium data points together with the data of Figure 5.12 are shown in Figure 5.13. These equilibrium points (above deposition limit) show the expected increase in shear stress with concentration unlike the data in the non-equilibrium situation (flow velocity below deposition limit).

![Graph showing shear stress vs. velocity](image)

**Figure 5.13** shear stress versus flow velocity including stationary flow.
5.3.6 Sedimentation velocity versus shear stress

The sedimentation velocity is plotted versus the measured shear stress in Figure 5.14 (left panel). In Figure 5.12 it was shown that the shear stress was not very dependent on the concentration in the settling regime. The relationship between the measured shear stress with (mean) flow velocity (for the mixture data) in Figure 5.12 can be approximated with a quadratic relation:

$$\tau_{cor} = \frac{f}{8} \rho \bar{u}^2$$  \hspace{1cm} (5.13)

Since the influence of the concentration is low the density of water is used in this equation. Using this relationship the sedimentation velocity can be plotted versus the calculated value for the average shear stress using the measured velocity instead of the measured shear stress. So instead of using the measured shear stress the calculated shear stress is used corresponding to the velocity for that test. The accuracy for the low velocity points is enhanced with this procedure. In the right panel of Figure 5.14 the sedimentation velocity is plotted versus the corrected shear stress.

![Figure 5.14 Sedimentation velocity versus shear stress \(d_{50}=125\ \mu m\).

5.3.7 Relation between the sedimentation velocity and shear stress for other sand

So far, the analysis was restricted to the tests performed on 125 \(\mu m\) sand. The same behaviour is observed for the other sands as is shown in Figure 5.15 and Figure 5.16. The sedimentation velocity increases with grain diameter as can be expected. The test results for all sands show the same dependency of the concentration. The sedimentation velocity does not vary much for concentrations between 15 – 30 %. This phenomenon was observed at the model sedimentation tests as well and was explained in Section 3.11. and 4.9.3.
Figure 5.15  Sedimentation velocity versus shear stress $d_{50}=150 \mu m$.

At the pilot tests the 30% concentration test results were very different from the results based on lower concentration. This phenomenon is not visible for the results shown in Figures 5.14 - 5.16. It can therefore be concluded that the observed inconsistent behaviour at the pilot tests was caused by the visual method used to determine the sedimentation velocity.

Figure 5.16  Sedimentation velocity versus shear stress $d_{50}=185 \mu m$.

5.3.8 Sedimentation velocity versus Shields parameter

The results of the different sands can also be plotted versus the Shields parameter. The Shield parameter is defined as:
\[
\theta = \frac{u_*^2}{\Delta gD} = \frac{\tau_{cor}}{\rho_s \Delta gD} = \frac{\tau_{cor}}{(\rho_s - \rho_w) gD}
\]  

(5.14)

The corrected shear stress is used. Since the density of water was used to compute that value, the same density will be used in the definition of the friction velocity \(u_*\).

![Graph showing sedimentation velocity versus Shields parameter for 125 and 150 \(\mu\)m sand.](image)

**Figure 5.17  Sedimentation velocity versus Shields parameter for 125 and 150 \(\mu\)m sand.**

The sedimentation velocity for the coarser sand was larger as can be expected. The sedimentation velocity is however equal to zero at approximately the same value of the Shields parameter as can be seen when the results of all sands are plotted in one figure. In Figure 5.18 the relative sedimentation velocity is plotted versus the Shields parameter. The relative sedimentation velocity is defined as:

\[
R = \frac{v_{sed}}{v_{sed,0}} = \frac{v_{sed}(\theta,c)}{\max(v_{sed}(\theta,c))}
\]  

(5.15)

The measured sedimentation velocity is normalised using the maximum sedimentation velocity for a certain concentration.
In the left and right panel the same data points are plotted. In the left panel a distinction is made with respect to the grain diameter, in the right panel the concentration is used. Within the total amount of data no systematic influence of the (mean) concentration or particle size on the sedimentation velocity can be observed. The results of the 270 μm sand show a lot of scatter. The high value of the sedimentation velocity for this sediment complicated an accurate determination of the sedimentation velocity.

It can however be concluded that the relative sedimentation velocity decreases more or less linearly with the Shields parameter. It was shown in Section 5.2.4 that the relative sedimentation velocity can be used to define a reduction factor for the settling flux.

\[ v_{\text{sed}} = R(\theta, c_b) = R(\theta, c_b) \frac{c_s w_s}{\rho_i (1 - n_0 - c_b)} \]  

(5.16)

A simple linear relation for the reduction factor as function of the Shields parameter can be obtained:

\[ R = \begin{cases} 
1 - \frac{\theta}{\theta_0} & \theta < \theta_0 \\
0 & \theta > \theta_0 
\end{cases} \]  

(5.17)

This relationship is used as the erosion-sedimentation boundary condition in the 2DV Model (see Chapter 7).

**5.3.9 Relationship between bed concentration and velocity**

The concentration of the settled bed is measured accurately using a radioactive concentration meter. The value of this concentration is plotted versus the flow velocity in Figure 5.19 for all sands.
Figure 5.19 Bed concentration versus velocity for all sands.

For all tested sand a very clear influence is observed of the flow velocity at which sedimentation takes place and of the packing of the grains in the bed. When the grains settle at zero velocity the loosest state of the bed is reached. With increasing velocity the sand bed becomes denser. An explanation is probably that a grain settling under flowing conditions can only stay inside the bed when it finds a more or less sheltered place between other grains. The latter results in a higher density of the sand bed. The data is fitted with a second order polynomial, except for 125 μm sand where a linear relation was used.

5.3.10 Pickup rate for high concentration

In par. 5.2.3 the pickup was computed from the measured sedimentation velocity of the pilot tests. In this Section the same procedure is repeated for the laboratory experiments of Delft University of Technology. The pickup rate can be computed again using equations (5.2), (5.3) and (5.4).
The following quantities are needed to compute the pick-up rate:
- The bed concentration \( (1-n_{b}) \).
- The near bed concentration \( c_b \).
- The sedimentation velocity \( v_{sed} \).
- The settling flux \( S \).

The bed concentration depends on the flow velocity and is approximated using the polynomial relations of Figure 5.19. For the near bed concentration in first instance the measured concentration above the bed before reducing the velocity was used, assuming that this value does not change much during the time that the sedimentation velocity was measured. The settling flux is computed using the near bed concentration, the settling velocity based on the \( D_{so} \) of the PSD and hindered settling according to Richardson and Zaki.

The results of this analysis are more reliable compared with the pilot tests, because in the pilot tests the concentration (both near bed and in the bed) were not measured. In Figure 5.20 the dimensionless pickup rate is plotted versus the flow velocity for the 125-μm tests. The difference between the two figures is the initial flow velocity \( u_0 \) (before the valve is closed), which was 2.6 m/s (left) and 3.6 m/s (right). The different symbols refer to the mean concentration before the velocity in the measurement section was reduced.

The back calculated pickup rate does not show a clear relationship with the concentration. Therefore, the near bed concentration was evaluated more in detail. It appeared that the near bed concentration during sedimentation was not always equal to the equilibrium value (present before flow velocity was reduced). Especially for high flow velocity, when the sedimentation velocity is low, the near bed concentration increases.
The pickup rate was computed again using the near bed concentration. The results are shown in Figure 5.21. Now a tendency can be observed that the pickup rate decreases with concentration. The pickup rate can be plotted as well versus the shear stress using the Shields value as shown in Figure 5.22. The Shields value is now based on the uncorrected shear stress $\tau_{\Delta p}$ to preserve all information (the drawback is more scatter and negative Shields values):

As a reference, the pickup function of Van Rijn (equation (2.25)) is plotted in the same Figure. It is clear that this function overestimates the pickup rate for most experiments.
This can be expected since this function was based on experiments with very low (almost zero) concentration and for Shields values smaller than unity.

In the above Figures the data points are identified using the mean concentration of the test. The settling flux is governed by the near bed concentration which may differ from the mean value. The pickup rate is therefore plotted as function of the Shields parameter in Figure 5.23 with the measured bed concentration as data label. The difference in pickup rate between low and high concentration is shown clearly in the figure for larger values of \( \Theta \). The difference is less clear for the data near the origin (low value of \( \Theta \)), which is to be expected since the pickup rate approaches to zero for all values of the concentration near the origin.

![Figure 5.23](image)

*Figure 5.23 Pickup as function of Shields parameter, 125 \(\mu\)m. Index is near bed concentration.*

In Figure 5.24 the pickup rate is plotted versus the velocity and Shields value for the 150 \(\mu\)m sand test results. The values using the same method are shown.
Figure 5.24  Pickup rate as function of velocity and Shields value, 150 μm. Index is near bed concentration.

Only the results for the high initial velocity are shown. The influence of the concentration on the pickup rate is shown clearly in this figure.

In Figure 5.25 the measured pickup rate is plotted versus the velocity and shear stress for the test performed on 185 μm sand. For these tests the value for the near bed concentration was more difficult to determine due to the larger sedimentation velocity. The difference in near bed concentration between the different tests is less compared with the previous analysed sands.

Figure 5.25  Pickup as function of velocity and Shields parameter, 185 μm. Index is near bed concentration.
Another interesting phenomenon can be observed in the right panel of Figure 5.25. The pickup rate for (very) high concentration (37 – 40 %) is approximately 0.02 – 0.025 for Shields values above 2. It becomes more or less constant for high concentration and shear stress.

![Graph showing pickup rate for different concentrations](image)

**Figure 5.26  Pick up as function of Shields parameter (270 μm).**

Finally in Figure 5.26 the shear stress for low and high initial velocity is plotted as a function of the Shields value for 270 μm sand. When the results of the different sands are compared, the following observations can be made:

- The pickup rate for the high initial velocity is slightly larger. The higher level of turbulence in the fluid most probably causes this after (partially) closing the valve.

- The pickup rate is lowest for the highest concentration for most sands. This might be caused by a “hindered erosion” effect (Winterwerp et al., 1992). Due to the presence of the high concentration in the suspension less space is available for eroded particles and consequently pickup is hindered. Another explanation could be that the calculated settling flux for this high concentration is too low, for instance due to a too large hindered settling effect. Nevertheless computations using different values for the hindered settling did not change the outcome.

- The pickup rate for the 270 μm is negative for Shields values below unity. This is most likely caused by an underestimated settling flux. To avoid these negative values a larger settling velocity is needed. The pickup rate only remained positive when the value of the settling flux was based on a grain size of 350 μm. Maybe the shape of the PSD is of importance. The 270 μm sand is less uniform, hence a relative large amount of coarse sand is present. Possibly also some segregation in the pipe was present already. Since the sedimentation velocity is based on the settling of the first two centimetres, the influence is large. With increasing grain size
the concentration profile becomes increasingly heterogeneous. This might be the case for the grain size as well.

5.4 Concluding Remarks

In this Chapter the influence of the bed shear stress on the sedimentation was investigated. Experiments executed at WL|Delft Hydraulics and Delft University of Technology were described and analysed. The influence of the bed shear stress on sedimentation is commonly taken into account using a pickup function. It was shown that the pickup rate depends on the near bed concentration. In a hopper net sedimentation takes place. In that case the influence of the bed shear stress can also be modelled by a reduction of the settling flux. This approach will be used for the 2DV model described in Chapter 7. In the future more research is needed to derive a pickup function valid for high concentration. This enables modelling of high concentrated sediment-water mixtures where net erosion takes place.

The tests executed with the 125 μm sand have shown that the reduction of the sedimentation velocity is small for a flow velocity below about 0.5 m/s. The measurements taken at the model hopper sedimentation tests have revealed that the maximum flow velocity in the density current is below this critical value most of the time. This implies that erosion did not play an important role during this test program. This was already argued in Section 4.8.6 and 4.9.3.
Chapter 6
One Dimensional sedimentation model (1DV)

6.1 Introduction
In Chapter 4 it was argued that apart from the inflow section, near the bed and near the water surface, the flow pattern in the hopper is basically one-dimensional in vertical direction (1DV).

![Diagram of flow pattern in hopper and 1DV Model]

Figure 6.1 Observed flowfield in a hopper and definition 1DV Model.

A density current flows over the bed and distributes the sediment over the length of the hopper. A part of the sediment does not settle in the bed and is transported in vertical direction by the vertical velocity component and turbulence. Assuming that the sediment influx is uniformly distributed over the length of the hopper the flow pattern of Figure 6.1 a) can be further schematised as indicated in Figure 6.1 b). Four different zones can be distinguished in Figure 6.1 b. From bottom to top:

1. Settled sand.
2. Area where sand is supplied to the model (simplification of the density current).
3. Suspension.
4. Overflow section (the overflow section normally present at one or two locations is uniformly distributed over the total surface).
In zone 2 the inflowing sandflux is prescribed. A proportion (depending on grain size and local concentration) settles into zone 1. The remainder is transported into zone 3. At the top (zone 4) the sand can escape into the overflow. The concentration development in zone 3 is described with the advection-diffusion equations for all fractions. The theory is outlined in the next Section.

Like all models this model is a simplification:

1. The suspension layer is supplied with a sand-water mixture with a concentration and discharge. It is assumed that the sandflux and discharge into zone 2 is the same as at the inflow location in the hopper. In reality this is not the case because due to entrainment the relative high concentrated inflowing mixture is diluted with lower concentrated mixture from the remaining hopper area.
2. All quantities (concentration, diffusion etc.) are assumed to be uniformly distributed over the length of the hopper.
3. The sand is uniformly distributed over the total surface of the hopper, so in fact an ideal inflow system is created (an infinite number of inflow points equally divided over the total hopper area). In reality the inflow is only located in a finite number of locations (in practice very often only at one location) and horizontal sediment transport must distribute the sand over the hopper surface. This horizontal transport is accompanied by a horizontal mixture velocity which, when above a certain threshold value, can reduce the sedimentation velocity as we have seen in the previous Chapter.

Simple models were analysed in Section 2.3. In these models the supply of sediment was at one side of the hopper in horizontal direction. The overflow was located at the other side. The sediment is transported horizontally by advection and vertically supported by turbulence. In the 1DV model proposed below the sediment is supplied just above the bed. Transport of sediment is in vertical direction by advection and diffusion (caused by turbulence). The mutual influence of the different grain sizes of the particle size distribution is included in the model.

### 6.2 Basic equations of the 1DV model

The vertical transport of sediment is zone 3 is described with the one-dimensional advection-diffusion equation. Using the grain-size distribution the incoming sediment flux can be distributed over the different fractions. The advection-diffusion equation for a fraction $i$ can be written as

$$\frac{\partial c_i}{\partial t} = -\frac{\partial}{\partial z}(c_i w_{xz,i}) + \frac{\partial}{\partial z} \left( \epsilon_{i} \frac{\partial c_i}{\partial z} \right) + \delta q_{i,i}$$  \hspace{1cm} (6.1)
In this equation is \( c_i \) the concentration and \( w_{z,i} \) the vertical velocity of a certain fraction. The vertical diffusion coefficient is represented with \( \varepsilon_z \). The equation includes a source term \( \delta q_{z,i} \) that is used to insert the sediment into the system. This source term is therefore only active at the cells located in zone 2 in Figure 6.1 b (\( \delta = 1 \) in zone 2 and \( \delta = 0 \) in other zones).

If the equation is solved for a mono-sized suspension (only one grain diameter present), the settling velocity for that grain size can be substituted for \( w_{z,i} \). The settling velocity of a grain is a function of the grain properties and the concentration. In general (Richardson and Zaki (1954)) this relationship is written as the product of a reduction function and the settling velocity of a single grain \( w_0 \):

\[
    w_i(c) = f(c)w_0
\]

(6.2)

In Section 3.5 several methods were described to extend this theory to a multi-sized mixture. Here the method of Mirza and Richardson (1979) is followed. It is assumed that every grain settles with a certain slip velocity \( v_{s,i} \) relative to the fluid velocity \( v_u \).

\[
    w_{z,i} = v_u - v_{s,i}
\]

(6.3)

The slip velocity is calculated according to Mirza and Richardson (1979) with (see Section 3.5):

\[
    v_{s,i} = w_0(1 - \bar{c})^{n-1}
\]

(6.4)

Where \( \bar{c} \) is the total volume concentration:

\[
    \bar{c} = \sum_{i=1}^{N} c_i
\]

(6.5)

To solve the advection-diffusion equation for all grain sizes the combined action of all grain sizes must be quantified. This can be done using the volume balance in vertical direction for both sand and water:

\[
    \sum_{i=1}^{N} w_{z,i}c_i + \left( 1 - \sum_{i=1}^{N} c_i \right) v_u = \frac{Q}{A} = w
\]

(6.6)

The simple relation for the vertical mixture velocity \( w \) is based on the assumption in the model that the discharge enters the hopper just above the bed. The volume balance (6.6) together with equations (6.3) and (6.4) forms a system of \( N+1 \) equations with \( N+1 \) unknowns. It follows that a simple relation can be derived from this system:

\[
    v_u = w + \sum_{j=1}^{N} c_j v_{s,j}
\]

(6.7)

Substitution in equation (6.3) leads to the following result:
\[ v_{z,i} = w + \sum_{j=1}^{N} c_j v_{s,j} - v_{s,i} \] (6.8)

Apart from the vertical bulk velocity \( w \), Smith (1966) already published this result. With the appropriate initial and boundary conditions the coupled set of advection diffusion equations can be solved numerically. The procedure followed is outlined in the next Section.

6.3 Numerical Procedure

The partial differential equation (6.1) is solved for every fraction using a finite difference scheme. A semi-implicit approach is used. The first term on the right hand side (advection) is treated explicitly with a first order upwind scheme or with central differences, the second term (diffusion) is treated implicitly with central differences. This leads to a tri-diagonal system of equations. The numerical procedure is as follows:

1. The initial values for the concentration of the different fractions must be known when the computation is started.
2. When at a certain time the concentration of all fractions is known, the right-hand side of equation (6.8) is known and the grain velocity for each fraction can be calculated.
3. Subsequently using the finite difference method the advection-diffusion equations for all fractions are solved independently for one time-step. This leads to a new concentration distribution on time \( t + \Delta t \).
4. Step 2 and 3 are repeated until the hopper is filled to certain level or for a certain total simulation time.

Apart from the initial conditions the following parameters and conditions must be known:

1. The boundary conditions.
2. The value of the diffusion coefficient \( \varepsilon_z \) as a function of height \( z \) and time \( t \).
3. The value of the vertical bulk velocity \( w \).
4. The value of the settling velocity \( w_0 \) of the different grain sizes (follows from the particle size distribution.)
5. The value of \( q_{s,i} \), the incoming sandflux per fraction (per unit height). To be determined from the PSD and bulk velocity \( w \).

A detailed description of the numerical procedure (Finite Difference Method) is beyond the scope of this thesis. Reference is made to Ferziger & Perić (1999). The boundary conditions and diffusion coefficient deserve however some extra attention.
6.4 Boundary Conditions

6.4.1 Source Term

The supply of sand into the system is implemented as a source term in the advection-diffusion equation in area 2. The inflowing sandflux for a certain fraction \( i \) per unit hopper area is:

\[ s_{in,i} = \frac{c_{in,i} Q}{A} \]  \hspace{1cm} (6.9)

Where \( c_{in,i} \) is the inflow concentration for fraction \( i \). When this flux is vertically distributed over a distance \( h_0 \) (see Figure 6.1 b), the source term in equation (6.1) for that fraction reads:

\[ q_{s,i} = \frac{s_{in,i}}{h_0} = \frac{c_{in,i} Q}{Ah_0} \]  \hspace{1cm} (6.10)

The value of \( h_0 \) is constant during a calculation. Using the grain-size distribution the total sand flux can be distributed over the different fractions. The cumulative particle size distribution (PSD) is used in the model to take the different grainsizes into account. If the cumulative PSD is presented with \( N+1 \) points \( N \) fractions are used.

![Figure 6.2 Close-up of grid near the bed.](image)

The diameter and concentration of fraction \( i \) is calculated as follows when the PSD is given with \( d_i \) and \( p_i \) (Figure 6.2)

\[ D_i = \frac{1}{2}(d_i + d_{i+1}) \hspace{1cm} c_{m,i} = \frac{c_m}{p_{i+1} - p_i} \]  \hspace{1cm} (6.11)

6.4.2 Water Surface

At the surface a special arrangement must be present to simulate the overflow. Normally the sandflux at the water surface is equal to zero. In this case the sandflux is prescribed near the water surface. At the surface:
\[-v_{z,j} c_i + \varepsilon_v \frac{\partial c_i}{\partial z} = -s_{\text{out}, j} \]  \hspace{1cm} (6.12)

At the water surface normally the sandflux is set to zero. In this case, the sandflux $s$ is prescribed at the surface to simulate the overflow.

\[s_{\text{out}, j} = c_i v_{z,j} \left( v_{z,j} > 0 \land Q > 0 \right) \]  \hspace{1cm} (6.13)

The two conditions must be included, since overflow only takes place when the mixture is discharged in the hopper, and to prevent the surface point from acting as a source term in case the vertical sand velocity is directed downwards. When both conditions are met the boundary condition equals a Neumann condition for the concentration.

### 6.4.3 Bed Boundary condition

The net sedimentation flux (depending on the near bed concentration) is calculated at the bed, and this amount is stored in the bed. During the calculations a fixed grid is used; the grid points are not moved with the bed variation. The problem with a moving bed and a fixed grid is solved as follows: At the start of the simulation the bed is situated at the lowest gridpoint. The values of the properties are defined in the grid nodes. The bed rises due to net sedimentation. After some time a situation as sketched in Figure 6.3 develops.

![Close-up of grid near the bed.](image)

**Figure 6.3 Close-up of grid near the bed.**

The bed is located somewhere between two grid nodes and is moving upward. The closest node below the bed is the bed node (lowest active gridpoint). The amount of sand moving into the bed vanishes from the suspension. This quantity is prescribed as a source in the lowest grid node. Therefore the boundary condition at the bed is:

\[-v_{z,j} c_i + \varepsilon_v \frac{\partial c_i}{\partial z} = -s_{\text{out}, j} \]  \hspace{1cm} (6.14)
The amount of sand of fraction $i$ moving into the bed is equal to:

$$s_{out,i} = v_{sed} \left( c_{i,bed} - c_{i,b} \right)$$  \hspace{1cm} (6.15)

Where $c_{i,bed}$ is the concentration of a fraction in the bed and $c_{i,b}$ the concentration of that fraction just above the sandbed (bed gridpoint). The sedimentation velocity reads (no influence of erosion):

$$v_{sed} = \frac{\sum_{lm} v_{z,l} c_{i,b}}{1 - n_0 - c_b}$$  \hspace{1cm} (6.16)

Where $c_{i,b}$ is the near bed concentration of a fraction $i$, and $c_b$ is the total near bed concentration. The concentration per fraction in the bed must be determined. From continuity it follows that:

$$\frac{c_{i,bed}}{c_{i,b}} = \frac{s_j}{s_{bed}} = \frac{\left( -v_{z,l} + v_{sed} \right) c_{i,b}}{(1 - n_0) v_{sed}}$$  \hspace{1cm} (6.17)

Which can be simplified to:

$$c_{i,bed} = \frac{\left( -v_{z,l} + v_{sed} \right) c_{i,b}}{v_{sed}}$$  \hspace{1cm} (6.18)

When the bed passes node $k+1$ in Figure 6.3 the lowest active gridpoint is shifted one place in upward direction (to $k+1$).

6.5 Vertical mixture (bulk) velocity

Because the mixture flows towards the bed of the hopper in the inflow section, an upward velocity is present in almost the complete zone were the mixture is in suspension. In the 1DV model this upward velocity $w = Q/A$ is present in the suspended zone (zone 3 of Figure 6.1). At the bed (boundary between zone 1 and 2) the vertical mixture velocity must be equal to zero. In zone 2 the vertical mixture velocity increases linearly from zero at the bed to $w = Q/A$ at the interface between zone 2 and 3.

6.6 Diffusion Coefficient

It is common to relate the diffusion coefficient to the turbulent eddy viscosity using the Schmidt number:

$$\varepsilon_c = \frac{v_c}{\sigma_t}$$  \hspace{1cm} (6.19)

Unlike the eddy viscosity, which is not a real fluid property since it depends strongly on the flow field, the Schmidt number only varies slightly within a flow and also slightly from flow to flow (Rodi, 1993). This conclusion even holds for flows where density
gradients are present (Uittenboogaard, 1995). Using the above relation the problem is shifted towards the determination of the eddy viscosity. This property is commonly computed using a turbulence model. Numerous turbulence models exist, from relative simple algebraic relations to complicated 2 (or more) (partial differential) equations like the well-known $k-\varepsilon$ model. Since the IDV model already strongly simplifies the actual flow pattern in the hopper it makes no sense to apply a complicated turbulence model. Instead the eddy-viscosity is assumed to be constant over the depth and the magnitude of this quantity is estimated using simple expressions. A possibility is the mixing length theory of Prandtl. The motivation for this approach is that turbulence is produced in the density current near the bed. The mixing length hypothesis reads:

$$v_e = \ell_m^2 \left| \frac{\partial u}{\partial z} \right|,$$

(6.20)

The mixing length increases with distance from the bed. When $h_0$ is the thickness of the density current the mixing length can be estimated as $0.09h_0$ (Rodi, 1993). Typical values measured from the (Model Hopper) experiments can be deducted from Figure 4.17. The thickness of the density current is of the order of $h_0 = 0.25$ m. The velocity gradient near the bed: $\frac{\partial u}{\partial z} = \Delta u / \Delta z = 0.25$ m/s/$0.1$ m = $2.5$ s$^{-1}$. This leads to a maximum value of the eddy viscosity (for the experiments) of $0.0013$ m$^2$/s.

Queck et al (1991) developed a 2DV model for sedimentation in clarifiers based on the assumption of constant density and viscosity. For the eddy viscosity they used as a length scale the height of the inflow section and as velocity scale the inlet velocity. So instead of turbulent production near the bed, the level of turbulence is related to the inflow properties.

$$v_e = \alpha u_0 d_{in} = \alpha \frac{Q}{B},$$

(6.21)

Where $B$ is the width of the hopper, $u_0$ the inflow velocity and $d_{in}$ the height (thickness) of the inflow zone. The value for $\alpha$ was based on flume tests and equal to 0.07. When this approach is applied on the hopper sedimentation tests a value for the eddy viscosity of approximately $0.0023$ m$^2$/s is found, which is of the same order of magnitude as the value computed using the mixing length theory.

Another approach is based on the parabolic eddy-viscosity distribution in river flow. The depth averaged the eddy-viscosity equals (using (2.39)):

$$\bar{v}_e = \frac{1}{6} \kappa u h = \frac{1}{6} \kappa \sqrt{\frac{f}{8} \bar{u} h} = \frac{1}{6} \kappa \sqrt{\frac{f}{8} \frac{Q}{B}},$$

(6.22)

This is the same approach as Camp in equation (2.15) and has the same appearance as (6.21) but leads to much lower values. For hopper sedimentation the last equation is less applicable since the flow is not distributed over the total flow depth but concentrated near the bed due to the density current behaviour.
6.7 Comparison between 1DV model and experiments

The 1DV model is compared with one-dimensional tests in a sedimentation column and the model hopper sedimentation experiments.

6.7.1 One dimensional sedimentation tests

The one-dimensional sedimentation tests (Klerk et. al., 1998, Runge & Ruig, 1998) are described in Appendix B. In a vertical column a sediment-water mixture was discharged and the sedimentation process was studied. A grid was placed in the column and by rotation of the column turbulence during sedimentation could be created. The results of these tests can be used to validate the 1DV model. Because the 1DV model includes the mutual interaction of the different fractions results are shown here of a test executed on a graded sand with a $D_{50}$ of 160 $\mu$m ($D_{10}$ is 85 $\mu$m, $D_{90}$ is 500 $\mu$m). The following values were used for the experiments and the numerical simulation:

Experiments:
- The particle size distribution of this sand is shown in Figure 6.4.
- Mixture level in the column was of 1.4 m.
- Initial concentration was 27 %.
- No (extra) turbulence was created by rotation of the column.

1 DV model
- The exponent in the hindered settlement function used in 1DV model is according to the values of Garside et al (1977), see equation (3.16).
- The level of turbulence in the column was low. Therefore the diffusion coefficient was set a low value of $5 \times 10^{-5}$ m$^2$/s and constant over the height of the column. This low value is of the order of magnitude of the self-diffusion (see Section 3.8).
- The particle size distribution was discretized in 10 different fractions in the computations.
The (total) concentration of the mixture was measured at twelve different vertical positions using electric conductivity probes (two-point probes, same type as were used for the hopper sedimentation experiments presented in Chapter 4).

In Figure 6.5 results from a sedimentation test are shown. Shortly after filling of the column the concentration is almost constant over its complete height. The sediment
settles which can be seen from the decreasing concentration in the upper part and increasing concentration in the lower part of the column where a bed is formed. The measured (symbols) and computed (continuous lines) concentration profiles are shown in Figure 6.5 at a 60 seconds interval during the settling process. The computed concentration in the suspension drops faster than measured. The difference is indicated with the arrows. For the computations a bed porosity of 0.4 was assumed, which agrees with the experiment for the lowest part of the settled bed (up to a height of 0.3 m). The measured bed concentration above 0.3 m height decreases gradually as can be seen in Figure 6.5. The segregation process during settling caused this phenomenon. All sizes are present in the suspension initially and in the lowest part of the bed all sizes can be found. After some time the coarsest particles have settled in the bed due to the larger settling velocity. Hence the difference in particle sizes present in the suspension decreases during time. The particle size distribution becomes more and more uniform. It is known that the porosity of the bed increases with the uniformity of the particle size distribution (Youd, 1973).

During the test samples were taken from the column at four different levels (0.3, 0.6, 0.9 and 1.2 m above the bottom of the column) at different times. The particle size distribution (PSD) was determined from these samples. The values of $D_{50}$ computed from the PSD’s are shown in Figure 6.6 as a function of time (symbols). The IDV model computes the concentration of each fraction as a function of space and time. This enables calculation of the value of $D_{50}$ in the model as a function of time on the same height as the sampling locations. The development of the computed $D_{50}$ (lines) as a function of time is compared with the measured values in Figure 6.6.

\[ D_{50} \ [\mu m] \]

\[ \text{time [s]} \]

**Figure 6.6** Calculated and measured $D_{50}$ as function of time at different vertical position. (No density correction).
It can be seen in Figure 6.6 that the development of the particle size distribution is simulated quite well with the model.

Figure 6.7  Calculated and measured concentration profile during one-dimensional settling of 160 μm graded sand. (With density correction).

The influence of the density correction is investigated by repeating the computations and taking the density correction according to Selim et al. (1983) into account.

Figure 6.8  Calculated and measured D50 as function of time at different vertical position. (With density correction).
The computed and measured concentration profiles are shown in Figure 6.7. When this Figure is compared with Figure 6.5 it is clear that the development of the concentration during time in the suspension is simulated more accurately when the density correction is used. The development of the particle size distribution (D_{50}) during time is however not properly predicted with the density correction as can be seen in Figure 6.8.

### 6.7.2 Hopper Sedimentation tests

Next, the numerical model is compared with the model hopper sedimentation tests analysed in Chapter 4. Two experiments are selected: A typical experiment with an average (for the test program) concentration and discharge (test 05).

<table>
<thead>
<tr>
<th>Particle Diameter [μm]</th>
<th>Cumulative Percentage [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>0</td>
</tr>
<tr>
<td>42</td>
<td>14</td>
</tr>
<tr>
<td>57</td>
<td>17</td>
</tr>
<tr>
<td>75</td>
<td>34</td>
</tr>
<tr>
<td>100</td>
<td>43</td>
</tr>
<tr>
<td>134</td>
<td>73</td>
</tr>
<tr>
<td>178</td>
<td>97</td>
</tr>
<tr>
<td>318</td>
<td>100</td>
</tr>
</tbody>
</table>

*Table 6.1 Particle size distribution used for simulation of test 06 and test 05.*

The other selected experiment was executed at the maximum possible (for the test program) sand flux (test 6). For both tests sand D_{50} = 105 μm. The PSD is schematised to 7 fractions. Table 6.1 shows the values used in the calculations.

Operational parameters during the experiments read:

- **Test 5**
  - Discharge: 0.099 m³/s
  - Density (average): 1310 kg/m³
  - Overflow level: 2.25 m
  - Water level at start: 1.25 m

- **Test 6**
  - Discharge: 0.137 m³/s
  - Density (average): 1420 kg/m³
  - Overflow level: 2.25 m
  - Water level at start: 1.25 m

The turbulent diffusion coefficient was kept constant over the height and equal to 0.0013 m²/s during the calculations (see. Section 6.6). The measured overflow concentration of Test 05 is compared with three different calculations in Figure 6.9:

- **Calc_1:** Multisized mixture (Table 6.1) with the density correction according to Selim et al. (1983).
- **Calc_2:** Multisized mixture without density correction.
- **Calc d_{50e}** Monosized mixture with a particle size equal to the d_{50} (105 μm) of the PSD.
The exponent of the hindered settling function was according to the values of Garside et al (1977), see equation (3.16).

![Figure 6.9](image)  *Measured and computed overflow concentration during test 5.*

The results from the mono-sized mixture underestimate the overflow concentration during the largest part of the loading time (which leads to a lower cumulative overflow loss). However, the final outflow concentration agrees with the measurements. The calculation with the density correction \(\text{calc}_1\) shows the best agreement with the experiments.

![Figure 6.10](image)  *Measured and computed overflow concentration during test 6.*

Test 6 (Figure 6.10) was performed at maximum (for the installation) incoming sand flux. Due to the high sand flux the measured concentration in the overflow remains almost constant during some time. The model reproduces this phenomenon very well (which only occurs at very high or low sand flux). At the end of the test the incoming sand flux decreases due to a lack of sand in the storage tank (the overflow losses during the test were high, so a lot of sand was needed). The overflow concentration drops sharply in response. This phenomenon is reproduced nicely by the model.
The computed (Calc1) and measured cumulative overflow losses are compared in Figure 6.11 and 6.12.

**Figure 6.11** Measured and computed cumulative overflow loss test 5.

**Figure 6.12** Measured and computed cumulative overflow loss test 6.

Both Figures show that the difference between computed and measured cumulative overflow loss is small as long as the complete particle size distribution is used in the calculation. When the complete PSD is approximated with one grain diameter the actual losses are underestimated.

### 6.7.3 Influence of the diffusion coefficient

The influence of the diffusion coefficient on the results of the IDV model can be made clear by variation of this parameter. This procedure is performed for test 06.

Figure 6.13 shows the computed overflow concentration and cumulative overflow loss for three different values of the diffusion coefficient $\epsilon$. It is clear that the development of the overflow concentration in time alters for different values of diffusion.
With increasing diffusion the concentration in the overflow increases at an earlier stage. The overall effect is however small as can be seen from the cumulative overflow loss. A factor ten difference in diffusion has a minor effect only. This indicates that advection is the dominant process over diffusion.

6.8 Discussion 1DV Model

The results of the 1DV model agree closely with the results of the experiments carried out when the mutual influence of the different grain sizes are included in the calculations. This can be achieved by using a multi-sized mixture in the 1DV model. The calculations were performed with and without the density correction according to Selim et al. Whether or not this correction should be included in the calculations is not clear. Sometimes the results improve, but not always. Beside, the method has the disadvantage that the results may depend on the number of fractions used to discretize the particle size distribution as is outlined in Appendix D.

It has not yet been established that the 1DV model will reproduce the process on full-scale as successfully as on model scale. In the 1DV model the influence of the horizontal transport, and the resulting bed shear stress that can decrease sedimentation velocity, was neglected. In Chapter 4 it was shown that the influence of the bed shear stress on sedimentation was negligible during the model tests due to scale effects. Neglecting reduction of sedimentation is therefore permitted on the scale of the model hopper sedimentation tests. If this still holds true for the prototype scale is not certain. Other limitations of the model were already mentioned in Section 6.1. In addition it can be mentioned that the influence of the loading and overflow system is not included in the 1DV model. These limitations motivated the development a two-dimensional model that will be described in the next Chapter.
Chapter 7

Two dimensional sedimentation model (2DV)

7.1 Introduction

This Chapter presents the two-dimensional (horizontal and vertical, 2DV) model that has been developed to simulate the sedimentation process in a hopper. One might question why a two-dimensional model should be developed when the results of the one-dimensional model already showed good agreement with the experiments. The motivation is that although the 1DV model predicts the results from the experiments quite well, it is unsure whether the comparison on prototype scale would be as favourable, due to the absence of horizontal transport in the 1DV model. The horizontal transport is accompanied with a horizontal velocity near the bed which can reduce the sedimentation velocity (Chapter 5). This phenomenon did not play a major role during the experiments due to scale effects.

7.1.1 Summary of the process conditions to be included in the model

A description of the sedimentation process inside the hopper is given in Section 4.7.2. The most important issues influencing the process to be included in the 2DV model are summarised below.

Non-stationary process

Due to the relatively high sedimentation velocity the filling time is of the same order of magnitude as the adaptation time of the suspended sediment distribution in the hopper. So in most cases the hopper is fully loaded before a stationary situation is established.

Non-Hydrostatic pressure distribution

The length and depth scales in a hopper are of the same order of magnitude. This implies that a correct representation of the flow in corners, which involves important vertical accelerations, is important. A hydrostatic pressure distribution (commonly used for estuarine models) is therefore not permitted since this leads to an unrealistic flowfield. The vertical and horizontal hydrodynamic pressure gradients must be taken into account.
Moving Bed
The interface between the settled sand and the mixture above the bed must be movable since it is the intention to fully load the hopper with sediment. Therefore a morphological model is required. The variation in flow depth is large, from maximum depth up to almost zero depth.

Moving water surface
The location of the water surface must be variable because of variation of the overflow level or filling of the hopper during the first phase of the loading process, when the water level is situated below the overflow level. The water level influences the sedimentation process in several ways. The most important influence is the entrainment of low concentrated mixture in the inflow section. When the flow depth increases the mixture in the inflow section must travel over a longer distance before the bed is reached. During this time more entrainment takes place. This leads to a lower concentration near the bed.

Influence of the particle size distribution
The results of the 1DV model have shown that not only the median particle $D_{50}$ of the sediment is important, but also the complete particle size distribution. The same influence is expected for the 2DV model and hence must be modelled.

Erosion / sedimentation boundary condition
The sedimentation velocity depends on the near bed concentration, the grain size and the bed shear stress. An accurate description of the erosion-sedimentation boundary condition is vital for the proper functioning of the model.

Influence of discharge system
A significant advantage of the 2DV model over the 1DV model is the possibility to include the influence of the inflow system in the model. The following variations are possible: number of loading points, location of the inflow point(s), inflow velocity and turbulence intensity.

Influence of overflow system
The influence of the overflow system on the process can also be taken into account with the 2DV model. The location and the number of overflows can be varied.
7.2 Limitations of the Model

The 2DV model also has some limitations:

- The hopper is simplified as a rectangular basin. In reality the lower part of hoppers have sloping side walls (Figure 2.4).
- The influence of the ship's motion is not included in the model. The pitching of the ship can be included, but it is not expected to have a large impact. Although the roll movement of the ship may have a reducing affect on sedimentation it cannot be modelled in the 2DV model. However in practice the effect is not important because the crew will always try to minimise the roll of the ship by sailing perpendicular on the waves during dredging as much as possible.
- The 2DV model is a flow model. The interaction between the flow and the settled sediment is modelled using an empirical description that holds as long as soil mechanical mechanisms are not dominant. This implies that the model may become inaccurate in cases where the loading of relative coarse sand is simulated. In such cases steep slopes develop inside the hopper and soil mechanical mechanisms (flow slides) influence the distribution of sand in the hopper. Fortunately, this type of sand is not relevant for modelling with the 2DV model since overflow losses are close to zero. (See Section 1.2), hence does not form a severe constraint for the 2DV model.

7.3 Basic Equations

7.3.1 Motivation of the mixture model

The equations used for the 2DV model are presented below. An Eulerian approach has been followed. Since the model deals with a mixture of sediment and water, a multiphase flow approach appears to be most appropriate. In such an approach, the continuity and momentum equation is solved for every phase. Every fraction must be considered as a single phase. To represent a Particle Size Distribution accurately this would lead to about ten different phases. A problem is the difficult (and often unknown) interaction between the different phases, which needs to be modelled. Despite the increased computing power, the multiphase approach is still often abandoned in industrial applications because of the long computing times, especially in the case of several dispersed phases. Complete multiphase simulations are also often numerically unstable and simplified forms of the inter-phase forces are often required (Taivassalo, 2001).

Various approximations have been developed in order to avoid full multiphase simulations. In civil engineering circumstances, concentration is often low and emphasis is given to modelling of the water flow. The transportation of solids is computed using a transport equation for the sediment after the flow field is solved without taking the influence of sediment on the flowfield into account. This involves a one-way coupling: the density variations are not used in the momentum equations. The
sediment is in fact treated as neutrally buoyant solvent. In a hopper sediment concentrations are too large to ignore density variations. The use of a complex numerical model for hopper sedimentation is only appropriate when relative fine sand is loaded. In cases where of coarse sands are being loaded almost all grains settle and a simple model will predict that outcome. As shown in Chapter 3, the Stokes number for fine sands is low, which means that the sand grains are following the mean flow. In that case the mixture model can be applied. The mixture model (or algebraic slip model) preserves the essential multiphase character of the flow, although it is an approximation of the full Eulerian model. It is based on the assumption of a local equilibrium between the continuous and dispersed phases, i.e. the difference between the (mean) particle velocity and (mean) flow velocity is assumed equal to the terminal slip velocity. (Taivassalo, 2001).

In this Chapter emphasis is on understanding the physics, so lengthy exact derivations are not supplied here; many textbooks are devoted to this.

7.3.2 Conservation of Mass and Momentum

For a fluid with constant viscosity the basic equation of motion are the Navier-Stokes (NS) equations:

\[
\frac{D(\rho \vec{u})}{Dt} = -\nabla p + \rho \vec{g} + \mu \nabla^2 \vec{u} \tag{7.1}
\]

The acceleration of a volume of fluid (left side of the equation) is the sum of the forces working on that fluid. In the right hand side of the equation three different contributions can be seen. The first term is a normal stress, the pressure gradient acting on the surface of the volume of fluid. The second term is the volume force due to gravity and the third term results from the shear stresses.

When the fluid is incompressible and non-diffusive, the following equation is valid:

\[
\frac{D\rho}{Dt} = 0 \tag{7.2}
\]

With this result the NS equation can be simplified to the following expression:

\[
\rho \frac{D\vec{u}}{Dt} = -\nabla p + \rho \vec{g} + \mu \nabla^2 \vec{u} \tag{7.3}
\]

It is usual to express the density and pressure as the sum of a hydrostatic and hydrodynamic contribution:

\[
p = p_0 + p_d \quad , \quad \rho = \rho_0 + \rho_d \tag{7.4}
\]
Substitution in (7.3) and division by \( \rho_0 \) leads to:

\[
\left(1 + \frac{\rho_d}{\rho_0}\right) \frac{D\vec{u}}{Dt} = -\frac{1}{\rho_0} \nabla p + \frac{\rho_0 + \rho_d}{\rho_0} \vec{g} + \frac{\mu}{\rho_0} \nabla^2 \vec{u} \tag{7.5}
\]

If the variation in density is small,

\[
\frac{\rho_d}{\rho_0} \ll 1 \tag{7.6}
\]

it is acceptable to neglect the variation of the density in the left-hand side of equation (7.5). This leads to an expression where the density variation only is present in the gravity term (Boussinesq approximation):

\[
\frac{D\vec{u}}{Dt} = -\frac{1}{\rho_0} \nabla p + \frac{\rho_0 + \rho_d}{\rho_0} \vec{g} + \frac{\mu}{\rho_0} \nabla^2 \vec{u} \tag{7.7}
\]

These simplifications were investigated to establish whether they could be used for the circumstances present in a hopper dredger.

With the Boussinesq approximation, the density variation must be compared with the density of water. From the experiments and practice we know that the density in the hopper, especially in the case of large overflow losses, can be as high as 1500-1600 kg/m\(^3\). So it must be concluded that the relative density variation is not small compared to unity and the Boussinesq approximation cannot be used.

The first approximation, the non-diffusive condition of the fluid, was also investigated. The density of the mixture depends on the sediment concentration.

\[
\rho = c\rho_s + (1-c)\rho_w \tag{7.8}
\]

Substitution in equation (7.2) leads to:

\[
(\rho_s - \rho_w) \left( \frac{\partial c}{\partial t} + u \frac{\partial c}{\partial x} + w \frac{\partial c}{\partial z} \right) = 0 \tag{7.9}
\]

The density of the sediment differs from water, so this equation can be simplified to:

\[
\left( \frac{\partial c}{\partial t} + u \frac{\partial c}{\partial x} + w \frac{\partial c}{\partial z} \right) = \frac{Dc}{Dt} = 0 \tag{7.10}
\]

This condition implies that the only transport mechanism of the sediment should be advection when (7.10) has to satisfied. It will be shown later (par. 7.4) that sediment is also transported by diffusion (caused by turbulence). The first approximation can therefore not be used. The NS-equation should therefore be solved in its original form (7.1).

So far the NS equation has been shown in its differential form. It will be shown later that the method used to solve the equations is the Finite Volume Method. (See par. 7.3.2) In that case it is more appropriate to formulate the NS-equation in its integral
form. This can be achieved by integration of the equation over a control volume $\Omega$. The outside area of the control volume (or perimeter in this case since a 2D problem is studied) is denoted with $S$. The horizontal momentum equation in integral form is:

$$\frac{\partial}{\partial t} \int_{\Omega} \rho u \, d\Omega + \int_{S} \rho u \vec{v} \cdot \vec{n} \, dS = \int_{S} \tau_{ij} \vec{i}_j \cdot \vec{n} dS - \int_{S} \rho \vec{f}_x \cdot \vec{n} dS$$

(7.11)

In the left-hand side of equation (7.11), the total derivative is split into a storage (unsteady) and the advection term. In the right hand side the stress term is still general; its relation with the flow field is not yet defined. In addition to the momentum equation the continuity equation is required to solve the system. For an incompressible fluid in conservative notation this equation becomes:

$$\int_{S} \vec{v} \cdot \vec{n} \, dS = 0$$

(7.12)

The relationship between the shear stresses and velocities must be known to solve this system of equations.

![Control Volume with definition of co-ordinate system](image)

*Figure 7.1  Control Volume with definition of co-ordinate system*

In Figure 7.1 a control volume is sketched. The top side of this volume is referred to as the north side. The shear stress on the top side (north side) of the control volume is elaborated. A linear relation exists between the shear stress and the velocity gradient for a Newtonian fluid.

$$\tau_{xz} = \rho \nu \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right)$$

(7.13)

Where $\nu$ is the kinematic viscosity of the fluid, and the product $\rho \nu$ is called the dynamic viscosity $\mu$. The shear stresses on the other surfaces of the control volume can be written likewise. All stresses can be expressed in terms of velocity gradients leading to a system of three equations (horizontal momentum, vertical momentum and continuity) with three unknowns: velocity components $u$ and $w$ and pressure $p$. A solution can be found in principle with the appropriate boundary conditions.
7.3.3 Turbulent Flow

The flowfield becomes unstable (turbulent) for high Reynolds numbers. Eddies are developing and the flow can only be described correctly if the smallest eddies are included in the calculations. The equations presented above have to be solved numerically. In case the shear stress is computed with (7.13) a very small grid size is required, which leads to huge computer memory demands and computational effort. Thus the so-called Direct Numerical Simulation (DNS) approach is not feasible for practical problems with large dimensions and large Reynolds numbers in the near future (Nieuwstadt, 1992).

However the time scale at which the turbulent fluctuations take place is frequently much smaller than the time scale of the boundary conditions. Velocities can therefore be averaged over a period that is relatively small to the changes in flow conditions (Rodi, 1993). The relevant quantities can be written in that case as the sum of a time averaged value and a turbulent fluctuation:

\[
\begin{align*}
    u &= \bar{u} + u' \\
    w &= \bar{w} + w' \\
    p &= \bar{p} + p' \\
    \rho &= \bar{\rho} + \rho'
\end{align*}
\]  

(7.14)

These expressions can be substituted into the NS equation leading to the so-called Reynolds equation. The complete procedure has not been reproduced here since it is rather lengthy and is readily available in many textbooks (e.g. Tennekes and Lumley, 1994). Here we focus on the advection term. On the north face of the control volume defined in Figure 7.1 (with length \(dx\) and height \(dz\)) the transport of horizontal momentum \(\rho u\) through the north face (length \(dx\)) by the vertical velocity \(w\) contribution is:

\[
\rho u w dx
\]  

(7.15)

Substitution of equations (7.14) in (7.15) leads to:

\[
\rho u w dx = (\bar{\rho} + \rho')(\bar{u} + u')(\bar{w} + w') dx
\]  

(7.16)

Averaging over time yields:

\[
(\bar{\rho} + \rho')(\bar{u} + u')(\bar{w} + w') dx = dx\left(\bar{\rho}\bar{u}\bar{w} + \bar{\rho}u'w' + \bar{\rho}u'w' + \bar{\rho}u'w'\right) +
\]

\[
dx\left(\rho'\bar{u}\bar{w} + \rho'u'\bar{w} + \rho'\bar{w}u' + \rho'u'w'\right) =
\]

\[
dx\left(\bar{\rho}u\bar{w} + \bar{\rho}u'w' + \bar{\rho}u'w' + \bar{\rho}u'w'\right)
\]  

(7.17)

The time averaged product of a mean and fluctuating component vanishes from the equation since the time averaged value of a fluctuating quantity is zero. The time averaged product of two or more fluctuating quantities is however not zero since these
quantities may be correlated. As a result five terms remain after time averaging. In
general (when a Boussinesq approximation is used) the last three terms are neglected. 
The remaining two terms are:

$$
\rho u w dx = dx \left( \rho uw + \rho u' w' \right)
$$

(7.18)

The first term is the transport of momentum due to the mean flow; the second term is a
turbulent contribution. This term can be interpreted as a shear stress. By moving this
term to the right hand side of the equation (7.11) the shear stress acting on the north
face of the control volume including the influence of turbulence can be expressed as:

$$
\tau_{x'} = \rho \nu \frac{\partial \bar{u}}{\partial z} - \rho u' w'
$$

(7.19)

The first term in the right hand side of (7.19) is the viscous shear stress (The second
term of equation 7.13 is omitted for clarity by taking \( \frac{\partial w}{\partial x} = 0 \)). The physical
meaning of the second term is an exchange of momentum due to turbulent eddies. A
vertical velocity fluctuation transfers a package of fluid with a different horizontal
velocity (as in the control volume) across the north face. This results in an acceleration
of the fluid in the control volume and acts like a shear stress. The origin however is
purely advection.

For clarity only the horizontal momentum equation is analysed and only one face of the
control volume is treated. For other faces and the vertical momentum equation the
stresses can be derived the same way. It can be shown that apart from shear stresses,
normal stresses appear as well. Take for example the advection through a vertical plane
of the control volume:

$$
\rho u w dz = \bar{\rho} (\bar{u} + u') (\bar{u} + u') dz = \rho dz \left( \bar{u}u + u'u' \right)
$$

(7.20)

These stresses originating from the momentum transfer by eddies are called the
Reynolds stresses in honour of the developer of this theory. The NS equation after time
averaging is called the Reynolds Averaged Navier-Stokes equation (RANS).

7.3.4 Eddy viscosity concept

Although expressions for the shear stresses are defined the system of equations cannot
be solved since new unknowns (the Reynolds stresses) have been introduced with the
Reynolds averaging. A method already proposed by Boussinesq in 1877 and still
frequently used is the eddy-viscosity concept. This concept assumes that analogous to
the viscous stresses in laminar flows, the turbulent stresses are proportional to the mean-
velocity gradients (Rodi, 1993). The turbulence is also supposed to be isotropic.
Application on the north side of the control volume:

$$
\bar{\rho} u' w' = -\rho \nu_e \frac{\partial \bar{u}}{\partial z}
$$

(7.21)
The quantity $\nu_e$ is called the eddy-viscosity. It is not a fluid property but a property of the flow field. Substitution in equation (7.19) results in the following expression:

$$
\tau_{xx} = \rho u' w' + u \rho' w' + w \rho' u'
$$

(7.22)

Initially, when elaborating equation (7.17), three terms were neglected. When a Boussinesq approximation is used this approach is permitted because the influence of density variation is only included in the gravity term and not in the advection terms. The Boussinesq approximation is not used here, therefore equation (7.17) is elaborated further. If we still neglect the last term (triple correlation) the turbulent contribution becomes:

$$
\tau_{xx} = \rho u' w' + u \rho' w' + w \rho' u'
$$

(7.23)

The eddy-viscosity concept can also be applied on the correlation between density and velocity. This approach is called the eddy-diffusivity concept:

$$
\rho' w' = -\frac{\nu_e}{\sigma} \frac{\partial \rho}{\partial z}
$$

$$
\rho' u' = -\frac{\nu_e}{\sigma} \frac{\partial \rho}{\partial x}
$$

(7.24)

Where $\sigma$ is the turbulent Schmidt-Prandtl number. This number takes into account that mass and momentum are exchanged differently by turbulence. The shear stress can now be written as:

$$
\tau_{xx} = -\left( \rho \nu_e \frac{\partial u}{\partial z} + u \nu_e \frac{\partial \rho}{\partial z} + w \nu_e \frac{\partial \rho}{\partial x} \right) = -\nu_e \frac{\partial (\rho u)}{\partial z} - \nu_e \frac{\partial \rho}{\partial x}
$$

(7.25)

In the approximation it is assumed that the Schmidt-Prandtl number is close to unity. For the situation in a hopper both the horizontal density gradient and the vertical velocity are low. Therefore the last term in equation (7.25) can be neglected and the remaining term is used in the model.

The problem has now been shifted towards the determination of the eddy-viscosity. A relative simple approach is the mixing length theory of Prandtl:

$$
\nu_e = \ell_m \left| \frac{\partial u}{\partial z} \right|
$$

(7.26)

In which $\ell_m$ is the mixing length. However the problem of how to determine this quantity remains. In shear flows near a boundary a reasonable approximation can be found, (like the distance to the wall) and the mixing length theory leads to satisfactory results. For general flow problems however this method is not suitable.
7.3.5 Turbulent closure with the $k$-$\varepsilon$ model

The RANS-equations can be solved when a relationship can be found between the flow field and the eddy-viscosity. The mixing length theory is too simple for general flow problems, so a more sophisticated approach is needed. The $k$-$\varepsilon$ model has been used with success for very different flow problems in the past. This model involves two transport equations that have to be solved alongside the NS equations.

The eddy-viscosity is proportional to the product of a velocity and a length scale characterising the (large-scale) turbulent motion:

$$\nu_t \sim U \ell$$  \hfill (7.27)

Another important property is the turbulent energy $k$, which is proportional with the square of the velocity:

$$k \sim \frac{1}{2}U^2$$  \hfill (7.28)

In this relationship $I_T$ is the turbulence intensity. For the dissipation of turbulent energy the symbol $\varepsilon$ is used. It can be shown that this quantity scales as follows with the velocity and length scale:

$$\varepsilon \sim \frac{U^3}{\ell}$$  \hfill (7.29)

This is called the Kolmogorov relation. Using the last three equations the following important relationship is derived:

$$\nu_t = c_\mu \frac{k^2}{\varepsilon}$$  \hfill (7.30)

In order to compute the eddy-viscosity the problem is now to determine the values of $k$ and $\varepsilon$. These quantities are obtained from the following transport equations (Launder & Spalding, 1974).

$$\frac{\partial k}{\partial t} + \frac{\partial (uk)}{\partial x} + \frac{\partial (wk)}{\partial z} = \frac{\partial}{\partial x} \left( \nu_t \frac{\partial k}{\partial x} \right) + \frac{\partial}{\partial z} \left( \nu_t \frac{\partial k}{\partial z} \right) + P + P_b - \varepsilon$$  \hfill (7.31)

$$\frac{\partial \varepsilon}{\partial t} + \frac{\partial (u\varepsilon)}{\partial x} + \frac{\partial (w\varepsilon)}{\partial z} =$$

$$= \frac{\partial}{\partial x} \left( \nu_t \frac{\partial \varepsilon}{\partial x} \right) + \frac{\partial}{\partial z} \left( \nu_t \frac{\partial \varepsilon}{\partial z} \right) + c_\varepsilon \frac{\varepsilon}{k} P + (1-c_\varepsilon) \frac{\varepsilon}{k} P_b - c_{2\varepsilon} \frac{\varepsilon^2}{k}$$

In the left hand side the rate of change of $k$ and $\varepsilon$ is given by the first term. The second and third term represent the transport of these properties by advection in horizontal and vertical direction respectively. The first and second terms in the right-hand side of the equation represent the diffusive transport in the horizontal and vertical direction. The diffusion coefficients are as usual the ratio of the eddy-viscosity and the Schmidt-
Prandtl numbers $\sigma_t$ and $\sigma_e$ respectively. The third and fourth terms give the production resulting from shearing of the fluid and buoyancy respectively.

The production term $P$ is the product of the second invariant of the deformation tensor with the eddy-viscosity. In tensor notation:

$$ P = \nu_e \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} $$

(7.32)

Written in full for completeness for two dimensions, $P$ becomes:

$$ P = 2\nu_e \left[ \left( \frac{\partial u}{\partial x} \right)^2 + \frac{1}{2} \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right)^2 + \left( \frac{\partial w}{\partial z} \right)^2 \right] $$

(7.33)

The influence of density gradients on the generation or damping of turbulence is modelled with the term $P_b$. This contribution is written as:

$$ P_b = \frac{g \nu_e}{\rho} \frac{\partial \rho}{\partial z} $$

(7.34)

On the application of this buoyancy term in the $\varepsilon$-equation the same procedure is followed as in Winterwerp (1999), based on Uittenbogaard (1995). For stable stratified flows the buoyancy term vanishes in the $\varepsilon$-equation ($c_{te} = 1$). In case of unstable stratification $c_{te} = 0$ is used, which represents small-scale turbulence production by Rayleigh-Taylor instabilities (Winterwerp, 1999). The values for the other parameters in the standard $k$-$\varepsilon$ model are shown in Table 7.1:

<table>
<thead>
<tr>
<th>$c_\mu$</th>
<th>$c_{1e}$</th>
<th>$c_{2e}$</th>
<th>$\sigma_k$</th>
<th>$\sigma_e$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1.0</td>
<td>1.3</td>
</tr>
</tbody>
</table>

Table 7.1 Standard coefficients of the k-$\varepsilon$ model.

The continuity and momentum equations together with the two transport equations form a strongly coupled system of equations. To solve the momentum equations the eddy viscosity must be known as a function of time and space. To compute the eddy-viscosity, $k$ and $\varepsilon$ must be known and these values are dependent on the velocity distribution.

7.3.6 The Re-Normalisation Group (RNG)- $k$-$\varepsilon$ model

In the previous Chapter the standard $k$-$\varepsilon$ was explained. Different variants of this model arise from determining the coefficients of the model. In some approaches the coefficient $c_{1e}$ is not constant, but a function of the ratio $\eta$ of production and dissipation of turbulent energy. This ratio is defined as:
\[ \eta = \frac{P}{\sqrt{c_\mu \varepsilon}} \]  \hspace{1cm} (7.35)

In the Re-Normalisation Group (RNG) \( k-\varepsilon \) model (Yakhot et. al, 1992) the coefficient \( c_{1\varepsilon} \) in the \( \varepsilon \) equation is now replaced with \( c_{1\varepsilon}^\ast \) calculated as follows:

\[ c_{1\varepsilon}^\ast = c_{1\varepsilon} - \frac{\eta \left( 1 - \frac{\eta}{\eta_0} \right)}{1 + \gamma \eta^3} \]  \hspace{1cm} (7.36)

Where \( \gamma = 0.012 \) and \( \eta_0 = 4.38 \). The other closure constants of the RNG-variant are shown in Table 7.2:

<table>
<thead>
<tr>
<th>( c_\mu )</th>
<th>( c_{1\varepsilon} )</th>
<th>( c_{2\varepsilon} )</th>
<th>( \sigma_k )</th>
<th>( \sigma_c )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.085</td>
<td>1.35</td>
<td>1.90</td>
<td>0.7179</td>
<td>0.7179</td>
</tr>
</tbody>
</table>

**Table 7.2 Coefficients of the RNG \( k-\varepsilon \) model.**

In Section 7.12 the two described variants of the \( k-\varepsilon \) model will be compared using different benchmark tests.

### 7.4 Sediment Transport

So far the attention has been focussed on the solution of the Navier-Stokes equation. If the density is constant in space and time and the boundary and initial conditions are known then the equations presented above are sufficient to find a solution for the velocity and pressure distribution as a function of space and time. However the density is not constant in a hopper, since the suspended sediment concentration in the hopper varies. Additional equations have to be solved to determine the distribution of sediment in the hopper. It is also important to know the sediment distribution because the sedimentation velocity and hence the filling of the hopper is directly related to the near bed concentration.

As with the 1DV model the grain size distribution is approximated with a finite number \( N \) of different grain diameters. The concentration of a certain fraction with grain diameter \( d_j \) is \( c_j \). The mixture density follows from the total concentration:

\[ \rho = \bar{c} \rho_s + (1 - \bar{c}) \rho_w \quad \bar{c} = \sum_{j=1}^{N} c_j \]  \hspace{1cm} (7.37)

The distribution of sediment concentration is calculated with the transport equation for each fraction. In differential form the transport equation reads:

\[ \frac{\partial c_j}{\partial t} + \frac{\partial (u_{\ast j} c_j)}{\partial x} + \frac{\partial (w_{\ast j} c_j)}{\partial z} = \frac{\partial}{\partial x} \left( \Gamma \frac{\partial c_j}{\partial x} \right) + \frac{\partial}{\partial z} \left( \Gamma \frac{\partial c_j}{\partial z} \right) \]  \hspace{1cm} (7.38)
Where:

- $u_{z,j}$: Horizontal velocity of the grains of fraction $j$.
- $w_{z,j}$: Vertical velocity of the grains of fraction $j$.
- $\Gamma$: Isotropic diffusion coefficient (sum of a turbulent and a molecular contribution. The latter contribution can be neglected in turbulent flows)

It is very important to realise that the numerical implementation of a transport equation in differential form is not necessarily conservative. So when a computer code is based on this relation the total quantity of sediment in the computational domain is not necessarily conserved. Material disappears (or appears) during the numerical process. This must be avoided, especially in case of the 2DV hopper sedimentation model where sand loss from the hopper is an important outcome. It is therefore more convenient to work in conservative or integral form, which forms the basis of the Finite Volume Method. The sediment transport equation in conservative form reads:

$$\frac{\partial}{\partial t} \int_\Omega c_j d\Omega + \int_s c_j \vec{v}_{z,j} \cdot \vec{n} \ dS = \int_s \left( \Gamma \nabla c_j \right) \cdot \vec{n} \ dS$$

(7.39)

The first term gives the rate of change of the concentration of a fraction, the second term represents the transport of that fraction by advection. In the right-hand side the diffusive transport is given. The turbulent contribution to diffusion is dominant for the type of flow encountered in a hopper. The diffusion coefficient $\Gamma$ is related to the eddy viscosity with the turbulent Schmidt-Prandtl number:

$$\Gamma = \frac{V_e}{\sigma}$$

(7.40)

Generally $\sigma$ is regarded as constant throughout the computational domain. It is however known that the mixing of sediment is affected by the concentration and grain properties. The difference between the diffusion of fluid momentum (eddy-viscosity) and diffusion of sediment can also be expressed as follows:

$$\Gamma = \beta \phi \nu_e$$

(7.41)

The $\beta$ factor takes into account that a sediment particle responds differently on turbulent velocity fluctuations than dissolved matter. Some investigators argue that this factor must be less than unity because sediment particles cannot fully respond to turbulent velocity fluctuations. Others claim that due to centrifugal forces sand particles are being thrown outside of eddies thereby increasing the mixing rate, resulting in $\beta > 1$.

Van Rijn (1984) analysed data of Coleman (1970) and found that the results could be described with the following relation which is always larger than unity:

$$\beta = 1 + 2 \left( \frac{w_r}{u_*} \right)^2 \quad 0.1 < \frac{w_r}{u_*} < 1$$

(7.42)
The factor $\phi$ takes into account the influence of the presence of the grains on the turbulent structure (damping in general). Two effects are dominant: damping due to the presence of a density gradient and increased dissipation due to an increased viscosity due to the presence of grains. The above relationships are mostly applied to the eddy-viscosity derived from clear water flow (a parabolic distribution). When a $k-\varepsilon$ turbulence model is used the influence of damping due to density gradients is already taken into account by the buoyancy terms in these equations. Since it is not clear if equation (7.42) is applicable for general flow problems, and the effect of $\phi$ is partly included in the turbulence model, the approach of Uittenbogaard (1995) is followed which shows that in free turbulence, even in fairly stratified conditions $\sigma = 0.7$ can be used. Hence equation (7.40) is used.

It should be noted that the velocity in the sediment transport equation is the grain velocity of that fraction. In the momentum equations the mixture (bulk) velocity is used. For every fraction a transport equation has to be solved. Except for very low total concentration, the transport of the different fractions is not independent. Different grain sizes will have a mutual influence as was already discussed in Chapter 6 where the 1DV model was described. The coupling between the different fractions is implemented like in the 1DV model.

### 7.4.1 Coupling of the fractions

In the horizontal direction it is assumed that the grain velocity is equal to the bulk (mixture) velocity of the suspension, so the horizontal component of the mixture velocity can be directly substituted in the sediment transport equations. In the vertical direction a slip velocity between the fluid velocity and grain velocity is assumed (algebraic slip model). Analogous to the 1DV model the following approach is followed to couple the grain velocities to the mixture velocity. Define:

- Vertical mixture velocity $w$
- Vertical fluid velocity $v_w$
- Vertical grain velocity $w_{z,j}$
- Slip velocity $v_{s,j}$

The definition of the slip velocity is:

$$w_{z,j} = v_w - v_{s,j} \quad (7.43)$$

The vertical mixture velocity is equal to:

$$w = \sum_{j=1}^{N} w_{z,j} c_j + \left( 1 - \sum_{j=1}^{N} c_j \right) v_w \quad (7.44)$$

Like in Chapter 3 this system of equations can be expressed as:

136
\[ w_{x,j} = w + \sum_{k=1}^{N} C_i v_{x,k} - v_{x,i} \] (7.45)

The slip velocity includes the influence of hindered settling (see Chapter 3) The mixture velocity \( w \) follows from the solution of the NS equations. Provided the concentration is known the slip velocity for every fraction can be calculated and consequently the grain velocities that are used in the sediment transport equations can also be calculated.

### 7.4.2 Influence of grain-grain interaction

In the previous section it was shown that advection and diffusion are responsible for transport of sediment in the 2DV model. In addition to these two mechanisms, grain-grain interactions can be important to support solid particles in suspension. The importance of grain-grain interactions can be characterised with the Bagnold number \( N \) defined with (Bagnold, 1954):

\[ N = \frac{\sqrt{\lambda \rho_s D^2}}{\eta_0} \dot{\gamma} \] (7.46)

Where \( D \) is the grain diameter, \( \eta_0 \) the dynamic viscosity of (clear) water, \( \dot{\gamma} \) the shear rate and \( \lambda \) the linear concentration defined with:

\[ \lambda = \left( \frac{c_{\text{max}}}{c} \right)^{1/3} - 1 \] (7.47)

Where \( c_{\text{max}} \) is the maximum possible concentration. According to Bagnold three regimes can be distinguished:

- \( N < 40 \) Macro viscous regime: viscous effects are dominant.
- \( 40 < N < 450 \) Transitional Regime
- \( N > 450 \) Grain inertia regime: grain collisions are dominant.

The three regimes can be visualised by plotting the shear rate as function of the concentration for Bagnold numbers equal to 40 and 450 respectively.

In Figure 7.2 the shear rate is plotted versus the sediment concentration. The lines are plotted for a grain size of 200 \( \mu \text{m} \). A value of 0.6 is assumed for \( c_{\text{max}} \). The figure shows that for this type of sediment the grain inertia regime is only of importance at a combination of a high shear rate and high concentration.
Figure 7.2  Different grain interaction regimes according to Bagnold.

The order of magnitude of the shear rate in the density current during the model hopper sedimentation test was estimated in Section 6.6. A value of 2.5 s⁻¹ was found. Figure 7.2 shows that this situation is located in the macro viscous regime. The presence of the solid particles increases the dynamic viscosity of the fluid. In the macro-viscous regime the shear stress has the following relationship with the shear rate according to Bagnold (1954):

$$\tau_{ex} = (1 + \lambda) \left(1 + \frac{\lambda}{2}\right) \eta_0 \frac{du}{dz}$$  \hspace{1cm} (7.48)

Where $\eta_0$ is the dynamic viscosity of the 'clear' (without sediment) fluid. Owing to the presence of sediment the effective viscosity of the fluid is increased by a factor which is a function of the linear concentration. The value of this factor is 100 when the volume concentration is 0.47. So only at a high volume concentration the effective viscosity is increased considerably, although these values are still small compared with the eddy-viscosity. The grain-grain interactions do not play an important role in the largest part of the suspension in the hopper, apart from the effect that the settling velocity of individual fractions is influenced by other fractions. This effect is modelled empirically by coupling of the fractions using the method of section 7.4.1. However a different situation occurs close to the bed.

At the bed, the shear rate can be quite large. Here the eddy-viscosity is relative low. Therefore settling of particles in the bed might be influenced by interactions between the grains. This effect is modelled empirically with the erosion sedimentation boundary condition based on the experiments described in Chapter 5.
7.5 Boundary Conditions

The hopper is schematised in Figure 7.3 as a rectangular domain.

![Diagram of computational domain](image)

*Figure 7.3 Definition of computational domain.*

Five different types of boundaries are required:
- Vertical walls
- Bed
- Water surface
- Inflow boundary
- Outflow boundary

7.5.1 Horizontal and vertical velocity

**Walls and Bed, low-Reynolds approach**

At each wall a thin viscous sub-layer is present followed by a buffer layer before we enter in the turbulent main flow. Very close to the wall the velocity component parallel to the wall is equal to zero. When such a “no-slip” condition is enforced at the wall, the complete turbulent boundary layer must be modelled using a very fine mesh close to the wall. Furthermore a low-Reynolds-number \( k - \varepsilon \) must be used close to the wall. In the low-Reynolds approach damping functions are used to modify the coefficients \( c_\mu \), \( c_{1\varepsilon} \) and \( c_{2\varepsilon} \) close to the wall. The other coefficients of Table 7.1 remain the same.

\[
\begin{align*}
    c'_\mu &= f_\mu \, c_\mu \\
    c'_{1\varepsilon} &= f_1 \, c_{1\varepsilon} \\
    c'_{2\varepsilon} &= f_2 \, c_{2\varepsilon}
\end{align*}
\]  \hspace{1cm} (7.49)

The following functions can be used according to Lam and Bremhorst (1981)

\[
\begin{align*}
    f_\mu &= \left(1 - e^{-0.0165R_\varepsilon}\right)^2 \left(1 + \frac{20.5}{Re_\tau}\right) \\
    f_1 &= 1 + \left(\frac{0.05}{f_\mu}\right)^3 \\
    f_2 &= 1 - e^{-Re_\varepsilon}
\end{align*}
\]  \hspace{1cm} (7.50)
With the Reynolds numbers:

\[ R_y = \frac{\sqrt{kY}}{\nu} \quad \text{Re}_\tau = \frac{k^2}{\nu \varepsilon} \quad (7.51) \]

In the last two equations \( Y \) is the normal distance to the wall and \( \nu \) is the kinematic viscosity of the fluid. Usual boundary conditions for the turbulent quantities are:

\[ k = 0 \quad \frac{\partial \varepsilon}{\partial Y} = 0 \quad (7.52) \]

The above relations cannot be used directly in the above formulation in a numerical method since both Reynolds numbers approach zero at the wall. To overcome these problems the function \( f_\mu \) is written as:

\[ f_\mu = \begin{cases} 
(1 - e^{-0.0165R_y})^2 \left(1 + \frac{20.5}{\text{Re}_\tau}\right) & R_y > 1.0 \\
(0.0165R_y)^2 \left(1 + \frac{20.5}{\text{Re}_\tau}\right) & R_y \leq 1.0
\end{cases} \quad (7.53) \]

Numerically it can still happen that \( f_\mu \) becomes relatively large close to the wall. In case \( f_\mu > 1 \) when \( \text{Re}_\tau < 1 \), \( f_\mu \) must be limited to 0.011. (Segal et al., 1996).

Furthermore the function \( f_2 \) should not be smaller than 0.01.

**Walls and Bed, wall function approach**

An alternative and widely adopted approach is the use of so-called “wall functions”. The logarithmic law of the wall is used to relate the velocity at the wall with the shear stress acting on the wall. For a hydraulic rough wall this relationship is:

\[ \frac{u}{u_*} = \frac{1}{\kappa} \ln \left( \frac{32z}{k_s} \right) \quad (7.54) \]

With \( \kappa \) as the Von Kármán constant and \( k_s \) the roughness height according to Nikuradse. The relation between the friction velocity and the wall shear stress is by definition:

\[ u_*^2 = \frac{\tau_b}{\rho} \quad (7.55) \]

Combining both equations leads to the following expression:

\[ \tau_b = \frac{\rho u_*^2}{\frac{1}{\kappa} \ln \left( \frac{32Y}{k_s} \right)^2} \quad (7.56) \]
In which $u_r$ is the velocity parallel to the wall in the gridpoint located at distance $Y$ from the wall (the closest gridpoint to the wall). Hence the velocity parallel to the wall is not equal to zero in the first gridpoint, but the fluid slips along the wall according to the above relationship. Using the definition of the friction velocity, these relationships can be expressed in terms of the velocity at the wall. For the hopper sedimentation model the wall function approach is chosen.

**Water Surface**

As mentioned in the introduction to this Chapter, the height of the water surface varies due to changing overflow levels and in the common situation where the initial water level in the hopper is lower than the overflow level. Several methods can be used to take account of the water surface. The first issue is whether to include surface waves in the model. For many applications in civil engineering (flows in estuaries and seas) this is certainly the case. For the hopper sedimentation model surface waves can only be of some importance in relation to ship motions. Since ship motions are not included in the model, surface waves have not been included either. Instead a so-called rigid lid approach has been followed. The water surface can be regarded as a super smooth horizontal plate covering the water surface in hopper. Depending on the total volume balance inside the hopper this lid is moved up and down. The vertical velocity of the surface $w_H$ follows from:

$$w_H = \frac{q_{in} - q_{out}}{L} \quad (7.57)$$

Where $q_{in}$ and $q_{out}$ are the specific in- and overflow discharge and $L$ is the length of the water surface. The vertical velocity is the boundary condition for the vertical velocity at the water surface. The overflow discharge depends on the difference between the overflow level and the level in the hopper:

$$q_{out} = C\sqrt{g} \left(h_h - h_o\right)^{3/2} \cdot \left((h_h - h_o) > 0\right) \quad (7.58)$$

The discharge coefficient depends on the geometry of the overflow crest. The following simple expression for the discharge coefficient provides satisfactory results for a 2D situation (Fox & McDonald, 1994)

$$C = 0.59 + 0.08\left(\frac{h_h}{h_o} - 1\right) \quad (7.59)$$

The effect of wind on the surface can be neglected, so for the horizontal momentum equation the shear stress at the surface is equal to zero.
**Inflow boundary**

The model is 2DV, hence it is assumed that the mixture enters the hopper over the total width. The vertical velocity is prescribed at the inflow section depending on the discharge and thickness of the jet. Usually the inflow is in the vertical direction through the water surface in the hopper. The vertical velocity is therefore prescribed over the width of the entering jet at the surface (see Figure 7.3). Discharge and jet thickness are input parameters for the model.

**Overflow (outflow) boundary**

The outflow boundary is only active when overflow is present when the hopper level exceeds the overflow level. In that case the outflow velocity is prescribed, and follows simply from the ratio of the overflow discharge and the difference between the hopper and overflow level.

\[
    u_{\text{out}} = \frac{q_{\text{out}}}{h_{\text{h}} - h_{\text{w}}} \quad (7.60)
\]

This is a simplification since a uniform velocity profile at the overflow crest is assumed. This is not correct and furthermore the water level on the crest is lower than the hopper level in the actual situation due to the free water surface. This simplification is a result of the rigid lid approach.

### 7.5.2 Boundary conditions for the $k-\varepsilon$ model

**Water surface**

For the boundary conditions for $k$ and $\varepsilon$ several possibilities can be found in literature. The normal gradient of the turbulent energy $k$ is very often put equal to zero at the surface (Adams, E.W. Rodi, W. (1989), DeVantier, Larock (1986), Stamou, Rodi & Adams (1989)).

\[
    \frac{\partial k}{\partial z} = 0 \quad (7.61)
\]

Another possibility is setting the value of $k$ to zero at the water surface (Bijvelds, 2001). For the dissipation $\varepsilon$ the normal gradient is put equal to zero by for instance Adams, E.W. Rodi, W., DeVantier, Larock (1986), Stamou, Rodi & Adams (1989):

\[
    \frac{\partial \varepsilon}{\partial z} = 0 \quad (7.62)
\]

Other possibilities are (Stansby, 1997):

\[
    \varepsilon = \frac{(k \sqrt{\varepsilon_p})^{3/2}}{0.07kh} \quad (7.63)
\]

Or the following comparable condition used by Siping Zhou & McCorquodale, 1992:
\[ \varepsilon = \frac{k^{3/2}}{0.43h} \]  

(7.64)

Where \( h \) is the flow depth and \( \kappa \) the Von Kármán constant. In the model several combinations of these possibilities were chosen, they did not significantly effect the results in case of hopper sedimentation.

**Boundary condition at the bed and side walls**

For the turbulent quantities the following boundary conditions are used at the wall in combination with the wall functions:

\[ k = \frac{u_r^2}{\sqrt{c_\mu}} \quad \varepsilon = \frac{u_r^3}{\kappa Y} \]  

(7.65)

Where \( Y \) is the distance to the wall.

**Inflow Boundary**

Turbulent quantities have to be prescribed at the inflow location. For the turbulent energy it is usual to prescribe \( (u_0) \) is the inflow velocity.

\[ k = \alpha u_0^2 \]  

(7.66)

The value of \( \alpha \) depends on the turbulence intensity in the inflowing mixture and can vary strongly depending on the application: Zhou and McCorquodale used \( \alpha = 0.2 \), Adams and Rodi, 1989 proposed \( \alpha = 0.03 \) and Zijlema, 1994 choose \( \alpha = 1.5 I_r^2 \), where \( I_r \) is the turbulence intensity, defined as:

\[ I_r = \sqrt{\frac{u'^2}{\bar{u}}} \]  

(7.67)

For the dissipation rate the following boundary condition is used at the inflow boundary:

\[ \varepsilon = \frac{c_\mu^{3/4} k^{3/2}}{\ell_m} \]  

(7.68)

The mixing length \( \ell_m \) is related to the width of the inflow section or inflowing jet. Most popular is the condition (Celik et al., 1985):

\[ \ell_m = c_\mu 0.5 d_{in} \]  

(7.69)

**Outflow Boundary**

For the turbulent quantities Neumann conditions are used:

\[ \frac{\partial k}{\partial x} = 0 \quad \frac{\partial \varepsilon}{\partial x} = 0 \]  

(7.70)
7.5.3 Boundary Conditions for the Sediment Transport equations

As with the momentum equations boundary conditions have to be supplied at the inflow- and outflow section, the vertical walls, water surface and the bed.

**Vertical walls and water surface**
At these locations the condition is that no sediment flows through these boundaries. So both the advection and diffusion of sand through these boundaries is set to zero.

**Inflow and outflow boundary**
At the inflow boundary the inflowing sandflux is prescribed. At the outflow a Neumann condition is used:

\[
\frac{\partial c}{\partial x} = 0
\]  

(7.71)

**Boundary condition at the sandbed**
At the beginning of the loading process the hopper is empty and the bottom of the ship is the bottom boundary of the domain. Due to the sedimentation process the bottom (bed) boundary for the flow moves up until the hopper is completely filled with sand. Two different approaches have been developed to deal with the moving sand surface:

**Keep the boundary at the original bed level**
In this case the boundary condition is simple. The sandflux through the bed must be zero. When overall sedimentation takes place the concentration in the bottom cell increases. When no special precautions are taken this value soon exceeds the maximum possible bed concentration, since the grain-grain interactions, which support the grains near the bed, are not included in the model. This unrealistic high concentration finally leads to numerical instabilities near the bed. This problem can be avoided when the settling velocity of the grains approaches zero when the total concentration reaches the bed concentration. The settling velocity is in that case modified with an expression like:

\[
w_{0,c} = (1 - e^{-A(c_b - c_r)}) w_0
\]  

(7.72)

Where \(c_b\) is the maximum bed concentration and \(c_r\) is the total near bed concentration. The sensitivity on the near bed concentration is determined by the factor A. However, a problem with this approach is that the influence of bottom shear stress on sedimentation has to follow from the behaviour of the solution near the sand surface. This only leads
to satisfactory results when all the relevant physics involved are present in the model formulation. However the physics of the transition from solid behaviour to fluid and vise-versa is not well known. Therefore another approach is followed which is more likely to produce practical results on short term.

**Move the flow boundary with the sand surface**

In this case the boundary of the flow calculation is the bed. The sedimentation velocity is low compared with the fluid velocity along the bed. For the flow calculation the bed is regarded as stationary at a certain time step. The new location at the bed for the next time step follows from the net sedimentation velocity, which is determined by the sum of the settling flux of all fractions. When the sedimentation is not reduced by the bed shear stress (stagnant flow condition), the expression for this variable becomes:

\[
\nu_{sed,0} = \frac{\sum w_{z,j}c_j}{1 - n_0 - \sum c_j}
\]  

(7.73)

The vertical grain velocity \( w_{z,j} \) near the bed is in general influenced by the bed shear stress. Sedimentation might be reduced. To quantify this reduction experiments were carried out (see Chapter 5, Closed Flume Tests). From the results the following approach was deduced. The effect of the flow velocity (or shear stress) can be shown by plotting a reduction factor (which is the ratio between the sedimentation velocity in presence of flow and the sedimentation velocity in absence of flow, the latter defined as \( \nu_{sed,0} \) ) versus the Shields value \( \theta \). In Figure 7.4 the measured reduction factor for all experiments is plotted for different sands for all concentration in the left figure and for different concentration in the right figure.

![Graph showing reduction of sedimentation velocity versus Shields parameter.](image)

**Figure 7.4** Reduction of sedimentation velocity versus Shields parameter.

145
Although a large amount of scatter is present, (especially for the relative coarse 270 μm sand) it is clear that the reduction factor decreases more or less linearly with the dimensionless shear stress. The following simple relation is therefore used for the reduction factor, as proposed in Section 5.3.8, equation (5.17)

$$R(\theta) = \frac{v_{sed}}{v_{sed,0}} = \begin{cases} 1 - \frac{\theta}{\theta_0} & \theta < \theta_0 \\ 0 & \theta > \theta_0 \end{cases} \quad (7.74)$$

The threshold value $\theta_0$ is approximately 4 – 5.
The expression for the sedimentation including the reduction by the bed shear stress reads:

$$v_{sed} = \frac{R(\theta) \sum w_{i,j} c_j}{1 - n_0 - \sum c_j} \quad (7.75)$$

For the sediment transport equations this condition leads to a prescribed sediment flux at the bed boundary because the Finite Volume Method is used to solve the transport equations (see Section 7.11).
The restriction of this method is that it cannot model eroding flows. For values of the Shields parameter above 4-5 the reduction factor will become zero and no sedimentation will occur in the model. However, in reality erosion will take place under these circumstances. To model eroding flows the bed boundary condition must be written as a pickup function.

### 7.6 Overview of the model

Now the three different parts of the model have been described the mutual relations can be shown in Figure 7.5. The different modules consist of the three different transport equations:

1. Transport of Momentum (2DV RANS)
2. Sediment Transport
3. Transport of turbulent quantities
In the 2DV RANS module the flowfield is solved. This can only be done when the spatial distribution of the eddy-viscosity and the density are known. The sediment transport equations and turbulence model supply these quantities respectively. The sediment transport and turbulence model both require the flowfield which is supplied by solving the momentum equations in the 2DV RANS module. The turbulence model and the sediment transport module exchange the eddy-viscosity and the density. From the eddy-viscosity the diffusion coefficient needed to compute the diffusive transport of sediment is calculated. The density gradient is used in the turbulence model to determine the dissipation (or production in case of unstable stratification) of turbulent energy.

7.7 Overview Numerical Methods

7.7.1 Introduction
The partial differential equations shown above must be solved to simulate the sedimentation process in a hopper. Due to the complexity of the system only a numerical approach is possible. Figure 7.5 shows that the total system of equations can be divided into three categories:
1. The Navier-Stokes equation together with volume conservation
2. Sediment Transport equations
3. Turbulence Model: transport equations for $k$ and $\varepsilon$.
A short overview of the different numerical methods is given below followed by analysis of the relative simple sediment transport equations. The description is continued with the more difficult treatment of the NS-equations for an incompressible fluid. Finally the implementation of the turbulence model is discussed.

7.7.2 Overview of numerical methods to solve partial differential equations

The summary below is very brief since numerous textbooks are available on this subject. For a general introduction the reader is referred to Anderson (1995 and Ferziger & Perić (1999). Another work, especially devoted to the Finite Volume Method, is the contribution of Versteeg (1995). In this book the treatment of the boundary conditions, very often underexposed by most authors, received the required attention. For general reference the reader is also referred to Fletcher (1991) and Hirsch (1990).

Finite Difference Method

This is the oldest method for numerical solution of partial differential equations (PDE's). The method is based on the PDE in differential form. The flow domain is, in most cases covered by a cartesian grid. At each grid point, the differential equation is approximated with a difference equation in which the variable in that gridpoint and neighbouring gridpoints appear as unknowns. The advantage of the method is the simplicity. A disadvantage of the method is however that conservation is not generally enforced. (Ferziger and Perić, 1999). Since the latter is not of great importance when solving the transport equations for the turbulent quantities, the method is used for these equations in this thesis, as will be shown in par. 7.10.

Finite Volume Method

The conservative form of the equations (for instance equation (7.39)) for the sediment transport) is used as the basis for this method. The flow domain is subdivided into a number of control volumes (CV's). At the centre of the CV's a grid point is located at which the variable is evaluated. Since we are dealing with transport equations the rate of change of the property is determined by the fluxes (advection and diffusion) at the control surfaces of the CV. Not all necessary quantities are known on the CV faces, so interpolation (from neighbouring CV's) is needed in most cases. As long as the interpolation practices are consistent (the same for the CV's sharing a boundary) the method is conservative. Another advantage is that the method can accommodate any kind of grid. In this thesis this method is used for both the sediment and momentum equations due to the conservative properties.

Finite Element Method

In this method the domain is divided into a finite number of control volumes or finite elements. Generally the elements are unstructured and in 2D usually triangles or
quadrilaterals. The difference with the Finite Volume method is that the equations are multiplied with a weight function before they are integrated over the entire domain. The solution is subsequently approximated by a shape function within each element. This approximation is then substituted into the weighted integral of the conservation law. The equations to be solved are derived by requiring the derivative of the integral with respect to each nodal value to be zero (leads to a minimum residual). (Ferziger and Perić, 1999). In this thesis this method has not been used.

7.7.3 Cartesian versus Boundary fitted grids

All methods mentioned above have in common that the equations have to be solved on a grid. The flow properties are therefore only known at discrete grid points. The procedure to transform the Partial Differential Equations (derived for a continuum) to algebraic equations to be applied on the grid points is called discretization in the literature dealing with numerical methods.

A Finite Difference Method is always implemented on a rectangular (cartesian) grid. A Finite Volume Method can in principle be applied on any grid, but it is advantageous to use a cartesian approach for this method especially when a staggered arrangement of variables is used (see par. 7.8.3). However in general the flow domain is not rectangular. The water surface can be considered horizontal on the length scale considered (for hopper sedimentation), but the bed does not necessarily coincide with the gridlines. Different approaches are possible. The first method is to use a cartesian grid and to adjust the cells near the bed. Another method is to fit the grid at the bed. In that case a boundary fitted non-orthogonal grid can be used. A third method is using grid transformation. By choosing an appropriate transformation the equations are solved on a cartesian domain in transformed co-ordinates (see Appendix C).

Although this transformation gives a good representation of a curved topography, the method has the disadvantage that due to truncation errors in the horizontal momentum equation, artificial flows develop when a steep bed encounters density gradients. These unrealistic flows can be partly suppressed when the diffusion terms are locally discretized in a cartesian grid. (Stelling & Van Kester, 1994, Stansby, 1997). However since both large density gradients and steep bed geometry can be present in a hopper, it was decided to develop the model in cartesian co-ordinates. This decision was made in 1999. Since that time a revival of cartesian CFD can be observed in the literature, driven by the increasing computational power and the fact that grid generation can be automated in these methods more easily.

7.8 Numerical Implementation Sediment Transport Equations

For the sediment transport equations and the momentum equations conservation is very important so the equations are solved using the Finite Volume Method. For the turbulence model conservation is less important, therefore the Finite Difference Method
is applied for this set of equations. A distinction must be made between the
discretisation of the equations in the interior of the computational domain and the
boundaries. To start with the implementation of the internal points are treated for the
sediment transport, momentum and turbulence equations. Finally the treatment of the
boundaries will be discussed for all equations.

7.8.1 Advection Diffusion Equation in Conservative Form

The advection-diffusion equation is summarised. The rate of change of the
concentration in a control volume $\Omega$ (left-hand side) is caused by the sum of the
advective flux and diffusive flux over the surface $S$ surrounding the control volume.

$$
\frac{\partial}{\partial t} \int_{\Omega} c_i d\Omega = - \int_S c_i \nabla_i \cdot \hat{n} \ dS + \int_S (\nabla c_i) \cdot \hat{n} \ dS
$$

(7.76)

When a Cartesian grid is used the computational domain is subdivided into rectangular
control volumes $\Delta x \cdot \Delta y \cdot \Delta z$. Since a two-dimensional approach is chosen $\Delta y = 1$ (unit
width) and all derivatives with respect to $y$ vanish. The value of the concentration is
defined in grid points at the volume centres (at points P,N,S,E and W in Figure 7.6).

![Figure 7.6 Cartesian Finite Volume grid.](image)

The fluxes are defined at the grid boundaries at locations n,s,e and w. Hence the grid
volume is $\Omega = \Delta x \Delta z$ and the surface is $S = 2\Delta x + 2\Delta z$. The partial differential equation
can now be discretized. For clarity one sediment fraction is regarded and indices $i$ are
omitted. The left-hand side is approximated with:

$$
\frac{\partial}{\partial t} \int_{\Omega} c d\Omega = \frac{\partial}{\partial t} (c_{p} \Delta x \Delta z) = \frac{c_{p}^{n+1} - c_{p}^{n}}{\Delta t} \Delta x \Delta z
$$

(7.77)

The time derivative is discretized using the first order Euler approximation. In the
equation is $c_{p}^{n}$ the concentration in the observed cell (in point P) at time step $n$, and
$c_{p}^{n+1}$ the concentration at the same location one time step $\Delta t$ later. A higher order scheme can be constructed using three time levels. (Ferziger & Perić, 1999):

$$
\frac{\partial}{\partial t} \int_{\Omega} c d\Omega = \frac{\partial}{\partial t} \left( c_{p} \Delta x \Delta z \right) = \frac{\Delta x \Delta z}{2 \Delta t} \left( 3u_{p}^{n+1} - 4u_{p}^{n} + u_{p}^{n-1} \right)
$$

(7.78)

This higher order scheme is not necessary, since the time step is small due to stability reasons owing to the coupling between sediment and momentum transport. The first order scheme is accurate enough.

The first term on the right side is the advective flux that is approximated on a Finite Volume (FV) grid in the following way:

$$
\int_{S} c \vec{v} \cdot \vec{n} \ dS = -\Delta z u_{e} c_{e} + \Delta z u_{w} c_{w} - \Delta x w_{e} c_{e} + \Delta x w_{w} c_{w}
$$

(7.79)

The fluxes are computed at the cell boundaries. When the values are determined in a consistent way, this scheme is conservative. The amount of material leaving one cell is always equal to the amount entering the adjacent cell. The same is true for the diffusive flux, which is computed as:

$$
\int_{S} \left( \Gamma \text{grad } c_{i} \right) \cdot \vec{n} \ dS =

-\Gamma_{w} \left( \frac{\delta c}{\delta x} \right)_{w} \Delta z + \Gamma_{e} \left( \frac{\delta c}{\delta x} \right)_{e} \Delta z - \Gamma_{x} \left( \frac{\delta c}{\delta z} \right)_{x} \Delta x + \Gamma_{z} \left( \frac{\delta c}{\delta z} \right)_{z} \Delta x
$$

(7.80)

In this equation the derivatives at the cell boundaries are written in a symbolic way using the $\delta$-operator. The background for this notation is that a certain approximation is needed to compute the gradient of the concentration at the cell boundaries (which is in reality a continuous function of time and space) using the values at the grid points. The above formulation for the fluxes is still general and expressed in values of variables at the boundary locations. For the actual implementation of the method these values have to be replaced by expressions based on values known at grid points. In equation (7.79), for example, values for $c_{w}$ etc. are needed, but have to be expressed in terms of concentrations available at the centres of the control volumes. A certain interpolation technique must therefore be used. These techniques form a subject on their own. Numerous publications and large parts of Computational Fluid Dynamics (CFD) handbooks are devoted to this subject (see for instance Hirsh, 1990). It goes far beyond the scope of this thesis to treat these methods in depth. Only the methods used for the subject model have been described.

The following quantities must be computed at the cell boundaries as shown above:

- Concentrations $c_{i}$, where $l = e, w, s, n$
- The velocities $u_{l}$
- Diffusion coefficient $\Gamma_{l}$

151
Concentration gradients in the x-direction on the w- and e faces and in the z-direction on n- and s faces. Some of the more popular interpolation practices will be highlighted in the next Section.

7.8.2 Interpolation Practices

Linear Interpolation

A straightforward approximation for the value at the Control Volume (CV) face is a linear interpolation between adjacent nodes. If a quantity $\phi$ is defined at locations P and W with values $\phi_p$ and $\phi_w$ respectively (Figure 7.6), then at the w-face of the CV the value $\phi_w$ follows from the following expression:

$$\phi_w = \lambda_w \phi_w + (1 - \lambda_w) \phi_p$$  \hspace{1cm} (7.81)

The factor $\lambda$ depends only on the horizontal grid spacing. If the horizontal CV size at location W and P is respectively $\Delta x_w$ and $\Delta x_p$ the interpolation factor is defined as:

$$\lambda_w = \frac{\Delta x_p}{\Delta x_p + \Delta x_w}$$ \hspace{1cm} (7.82)

Similar expressions can be derived for the other faces. It can be shown that the truncation error of this scheme is proportional to the square of the grid spacing, hence it is second order accurate (Fletcher, 1991). Since the interpolation corresponds to the central difference approximations of the first derivative in finite difference methods the above interpolation is called the Central Difference Scheme (CDS). Although accurate this scheme has one major drawback: it may produce oscillatory solutions. This may even lead to negative values of the transported property. This is physically unrealistic and can as well lead to serious stability problems when solving the $k-\varepsilon$ equations, as will be discussed later.

Upwind Interpolation

In the upwind interpolation scheme (very often called Upwind Difference Scheme, UDS) the value at the face centre is approximated with the value at the node upstream of that point. So in that case $\phi_w$ is computed using:

$$\phi_w = \begin{cases} 
\phi_w & \text{if } u_w > 0 \\
\phi_p & \text{if } u_w < 0
\end{cases}$$ \hspace{1cm} (7.83)

This scheme never yields oscillatory solutions (or negative values) and is therefore often called a positive (or monotone) scheme. It is however only first order accurate and is numerically diffusive, especially when the flow is oblique to the grid.
Higher order positive schemes
Much work has been done to develop interpolation techniques, which have positive (monotone) characteristics and are second order accurate. These schemes are called Total Variation Diminishing (TVD) Schemes (Hirsch, 1990 has an extensive overview). For the sediment transport equation of the hopper model a MINMOD scheme has been used for the advection terms (Hirsch, 1990).

7.8.3 Approximation of the diffusive flux
In equation (7.80) a symbolic expression was shown for the diffusive flux at the west face of the control volume. The gradient of the concentration at the west face can be approximated using the adjacent grid values:

$$\Gamma_w \left( \frac{\delta c}{\delta x} \right)_w \Delta z = \Gamma_x \frac{c_p - c_w}{x_p - x_w} \Delta z$$  \hspace{1cm} (7.84)

Staggered grid arrangement
In equation (7.80) and equation (7.84) it can be seen that apart from the concentration, other quantities (velocity for the advection and the diffusion coefficient) must be also be determined at the cell faces. When these quantities are defined at the same locations (grid points) as the concentration the above described interpolation technique can be used to approximate the values at the faces. In that case all quantities share the same grid. This is called a collocated grid. It is however not obligatory that all quantities share the same grid. It can even be very advantageous to define other quantities (for instance horizontal and vertical velocity) at other locations, as will be shown in Section 7.10. This is called a staggered grid arrangement (Harlow & Welch, 1965). In Figure 7.7 the arrangement chosen for the 2DV model is shown.

![Staggered grid arrangement](image_url)

"Figure 7.7 Staggered grid arrangement."

In this Figure four different grids in a double staggered arrangement are shown (dots, stars, triangles, rectangles).
The different grids accommodate the following quantities:
Star: horizontal velocity
Rectangle: vertical velocity
Dots: Concentration, density and pressure.
Triangles: Eddy viscosity, $k, \varepsilon$

**Implicit or Explicit**
In the previous Section it is shown how the transport equation can be discretized. The partial differential equations were reduced to a system of algebraic equations where the value of a quantity in a certain gridpoint can be expressed in the terms of neighbouring values. Substitution of (7.77), (7.83) and (7.84) leads to the following expression for every gridcell:

\[
\frac{c_p^{n+1} - c_p^n}{\Delta t} \Delta x \Delta z = A_p c_p + \sum_{l=N,S,E,W} A_l c_l + Q_p
\]

(7.85)

In which the coefficients of $A'$ depend on the grid spacing, velocity field and the diffusion coefficient: $A_l = f(u, w, \Delta x, \Delta z, \Gamma)$. The term $Q$ represents source terms and or expressions depending on not-adjacent gridpoints resulting from higher order methods.

An important issue remains to be addressed. In the right-hand side of the above equation there yet an indication at what time step ($n$ or $n+1$) the values of the concentration are used. If all values are used from the previous time step $n$ there is only one unknown in equation (7.85) and the value of $c_p^{n+1}$ can explicitly be expressed in the known concentration values of the previous time step. Therefore this is called an explicit scheme. Such a scheme is very easy to implement, but most of the time it is only stable at a very small time step. Due to these stability problems in practice an implicit method is favoured. In a (fully) implicit approach the values in the right-hand side of (7.85) are taken at the new (not known yet) ($n+1$) time step. The above equation can in that case be written as:

\[
A_p c_p^{n+1} + \sum_{l=N,S,E,W} A_l c_l^{n+1} = Q_p^n
\]

(7.86)

When the grid consists of $N$ grid cells, we have a linear system of $N$ equations (every cell provides an equation) with $N$ unknowns that must be solved for every time step. Since every cell only influences neighbouring cells, the system matrix is, apart from a small band along the diagonal, almost completely filled with zeros. Very efficient algorithms (solvers) exist to solve for these so-called sparse matrices.
7.9 Numerical implementation of the Navier-Stokes equations

7.9.1 Introduction

In the previous Chapters the numerical implementation of the transport equation for sediment was discussed. Some basic aspects regarding grids, discretization and interpolation techniques were introduced. In this Chapter the analysis is continued with the implementation of the NS-equation on a FV grid. Although this equation can be regarded as a transport equation of momentum the numerical implementation is more complicated. The basic problem is highlighted below and the analysis continues with a review of different possible solutions and the method chosen for the 2DV model.

7.9.2 Basic problem with NS-equation for an incompressible fluid

Solving the NS-equation for an incompressible fluid is much more difficult then solving a transport equation for a scalar quantity. For clarity the NS-equation is analysed in differential form for a constant density and viscosity (equation (7.87)):

\[
\frac{D(\rho u)}{Dt} = -\frac{\partial p}{\partial x} + \nu \nabla^2 (\rho u)
\]

\[
\frac{D(\rho w)}{Dt} = -\frac{\partial p}{\partial z} + \nu \nabla^2 (\rho w) - g
\]  

(7.87)

The general formulation of the transport equation for a general scalar quantity (equation (7.88) for constant diffusion) reads:

\[
\frac{D\phi}{Dt} = \Gamma \nabla^2 \phi
\]

(7.88)

When (7.87) and (7.88) are compared it is clear that the NS-equation is a transport equation for momentum in the horizontal and vertical direction with (apart from the acceleration due to gravity) the pressure gradient as an extra term. The pressure term causes the main difficulty. Even with a relative simple explicit approach where the spatial derivative in the right-hand side is based on the information of the previous time step we still need the information of the pressure gradient to update the velocity to the next time step. No extra equation is available to explicitly calculate the pressure when the fluid is incompressible. The only additional equation is continuity:

\[
\frac{\partial u}{\partial x} + \frac{\partial w}{\partial z} = 0
\]

(7.89)

In which the pressure is not present either, so this is not directly of any help. The basic problem is that two equations have to be solved (for horizontal and vertical momentum), which contain an unknown pressure gradient and the resulting velocities have to satisfy the continuity equation (which does not contain a pressure term).
7.9.3 Possible Solutions

Hydrostatic pressure distribution

When the horizontal dimensions of the flow are large compared to the water depth a possible approach is the hydrostatic pressure distribution. Vertical acceleration of the flow is neglected and the vertical momentum equation reduces to the following simple expression:

$$\frac{\partial p}{\partial z} = -\rho g$$  \hspace{1cm} (7.90)

The pressure only depends upon the distance from the water surface, and the pressure gradient can be computed if the water level is known. How to update the water level as a function of time is not addressed here. The hydrostatic approach is not used for the 2DV model because the depth and length of the flow domain is of the same order. In that case the vertical momentum transport cannot be neglected anymore, otherwise a very unrealistic flowfield results.

Streamline Vorticity approach

Another approach is elimination of the pressure term by differentiation of the horizontal momentum equation with respect to z and the vertical momentum equation with respect to x. After this procedure both equations contain the same pressure term $\partial^2 p / \partial x \partial z$ that can be eliminated by subtraction of the equations. The resulting lengthy expression contains many mixed derivatives, but can be simplified considerably by introducing the streamfunction $\psi$ and vorticity $\omega$ as new variables. These two quantities are defined as:

$$\frac{\partial \psi}{\partial z} = u \quad \frac{\partial \psi}{\partial x} = -w$$  \hspace{1cm} (7.91)

$$\omega = \frac{\partial w}{\partial x} - \frac{\partial u}{\partial z}$$

Substitution of these quantities in the lengthy expression, and application of the continuity equation, leads to the following system of equations (Ferziger and Perić, 1999):

$$\rho \frac{D\omega}{Dt} = \mu \left( \frac{\partial^2 \omega}{\partial x^2} + \frac{\partial^2 \omega}{\partial z^2} \right) = \mu \nabla^2 \omega$$  \hspace{1cm} (7.92)

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial z^2} = -\omega$$

The number of dependent variables has been reduced with this method and the familiar transport equation appears for vorticity together with a Poisson equation for the
streamfunction. The numerical procedure in this case can be as follows (Quek et al., 1991).

1. Update the vorticity by solving the transport equation as in par. 7.8.1
2. Using the new vorticity update the Poisson equation for the stream function.
3. From the stream function the new velocity components can be calculated using (7.91)
4. The new velocity is used to update the boundary conditions for the vorticity.
5. Repeat above procedure.

Although the problem with the pressure gradient has been solved with the above approach other difficulties arise:

- Neither vorticity nor its derivatives at the boundary are known in advance.
- At sharp corners the vorticity is singular and special care must be taken to compute the values at these locations.
- The extension to three-dimensional flows is difficult.
- It is difficult to deal with variable fluid properties like eddy-viscosity and density.

The streamline-vorticity approach is not applied for the hopper sedimentation model mainly due to the difficulty to include variable density (and hence density currents) in the numerical model.

The Pressure Correction Method

In this Section the method used in the present model is described. The NS equation in its integral form is repeated. (only for the horizontal velocity component).

\[
\frac{\partial}{\partial t} \int_{\Omega} \rho u d\Omega + \int_{S} \rho u \cdot \vec{v} \cdot \vec{n} \cdot dS = \int_{S} \tau_{ij} \cdot \vec{n} dS - \int_{S} p \cdot \vec{i} \cdot \vec{n} dS
\]  

(7.93)

The first term in the left-hand side is discretized the same way as was done for the sediment transport equation:

\[
\frac{\partial}{\partial t} \int_{\Omega} \rho u d\Omega = \frac{(\rho u)^{n+1} - (\rho u)^{n}}{\Delta t} \Delta x \Delta z
\]  

(7.94)

With this formulation the density must be known on the time step \( n+1 \), which cannot be determined unless the flowfield is known. This problem is addressed below in Section 7.9.6.

7.9.4 Turbulent Shear Stresses

Previous Chapters discussed how the Reynolds-averaging and the eddy-viscosity concept were applied. The shear stress term (first term in right hand side) in Finite Volume is written as:
\[
\int_{S} \tau_{ij} \vec{i}_j \cdot \vec{n} dS = \Delta z (\tau_{xx})_e - \Delta z (\tau_{xx})_w + \Delta x (\tau_{zx})_n - \Delta x (\tau_{zx})_s \quad (7.95)
\]

The shear stress is written in terms of the product of the eddy viscosity and velocity gradient. For example for the north face:

\[
\Delta x (\tau_{zx})_n = \Delta x \left( \rho v_N \frac{\delta u}{\delta z} \right)_n = \Delta x \rho v_{e,s} \frac{u_{e}^{n+1} - u_{e}^{n}}{\Delta z_n} \quad (7.96)
\]

This is the most important part of the shear stress on the north face and is dealt with implicitly. For the other faces similar expressions can be derived.

### 7.9.5 Advection

The advection term requires special treatment. This term is discretized as:

\[
\int_{S} \rho u \vec{v} \cdot \vec{n} dS = -\Delta z (\rho uu)_{e} + \Delta z (\rho uu)_{w} - \Delta x (\rho uw)_{n} + \Delta x (\rho uw)_{s} \quad (7.97)
\]

Where \( w,e,s \) and \( n \) are the boundaries of the control volume for the horizontal velocity (see Figure 7.8). When an implicit method is used there are problems with this expression since it is non-linear. The general approach is linearizing these terms by a product of old and new time or iteration level. For example the flux at the west face of a CV is written as:

\[
(\rho uu)_{w}^{n+1} = (\rho u)_{w}^{n} u_{w}^{n+1} \quad (7.98)
\]

Note that again the interpolation problems as discussed in par. 7.8.2 arise. In this case the horizontal velocities at the faces have to be determined from adjacent grid points using a CDS, UDS or higher order scheme. At the north and east face a vertical velocity component is needed as well as can be seen from equation (7.97).

![Figure 7.8 Control volumes for the horizontal (l) and vertical velocity (r).](image)
These components can straightforward be computed from the gridpoints located at the corners of the u-Control Volume. (for the north face location nw and ne).

7.9.6 Pressure gradient

Except for the problematic pressure term all contributions have now been discretized. The horizontal momentum equation is now written as:

\[ A_p^{u} \bar{u}_p^{n+1} + \sum_{i=N,S,E,W} A_i^{u} \bar{u}_i^{n+1} = Q^u - Q_p^{n+1} \] \hspace{1cm} (7.99)

This is a linear system of equations. Every CV provides an equation and the number of unknown velocities is equal to the number of CV's. In the right hand side the term \( Q^u \) gathers all explicit contributions. The pressure is split into hydrostatic and hydrodynamic parts. The hydrostatic part is treated explicitly. The hydrodynamic pressure term is represented with \( Q_p^{n+1} \). This term is discretized as follows for the horizontal momentum equation:

\[ -\int_S \bar{r}_s \cdot \bar{n} dS = - (\Delta z p_w - \Delta z p_e)^{n+1} \] \hspace{1cm} (7.100)

Due to the staggered arrangement of the variables the position for the pressure grid point is right on the face of the control volume for the horizontal velocity. Again the problem is encountered that the pressure gradient is unknown for time step \( n+1 \). The pressure field has to be corrected for the new time step, otherwise continuity cannot be assured. Patankar (1980) has developed a solution for this problem as outlined below. The first step is to guess a pressure field. The pressure field of the previous time step is a good starting point. With this guessed pressure field equation (7.99) and likewise the vertical momentum equation can be solved. When the guessed pressures are denoted with \( p^* \) the resulting horizontal and vertical velocities can be written as \( u^* \) and \( w^* \). Generally, this velocity field does not satisfy the continuity equation. The task is now to find corrections for the pressure and velocity components in order to satisfy both the momentum and the continuity equation for the next time step. These quantities have to be obtained from the following relations:

\[ p^{n+1} = p^* + p' \]
\[ u^{n+1} = u^* + u' \]
\[ w^{n+1} = w^* + w' \] \hspace{1cm} (7.101)

The first step is to find the pressure corrections. These terms can be computed from a discretized form of a Poisson Equation, the so-called pressure correction equation (Patankar, 1980):
\[ a_p p_i' - \sum_{l=NSWEW} a_i p_i' = \Delta z \left( \frac{(\rho u^*)_w - (\rho u^*)_s}{\Delta x} \right) + \Delta x \left( \frac{(\rho w^*)_s - (\rho w^*)_e}{\Delta z} \right) \] (7.102)

This equation forms a linear system of equations. In the left-hand side the unknown pressure corrections can be seen. The right hand side is known since these quantities followed from solving the system with the guessed pressure field.

This relation follows from application of the continuity equation on a scalar control volume (see Figure 7.9). The coefficients \( a_p, a_w \) etc. depend on the grid spacing and the discretization of the momentum equations.

\[ \begin{array}{cccc}
\bullet & p_i' & \bullet \\
\bullet & u_i' & \bullet & p_i' \\
\bullet & \Delta x \\
\bullet & \Delta z \\
\end{array} \]

Figure 7.9  Control Volume for pressure correction equation.

Using the pressure corrections the velocity corrections can be computed and the guessed pressure en velocity field can be updated. This flowfield satisfies continuity, but generally not the momentum equations. Therefore the above procedure can be repeated with the corrected pressure field. This procedure is repeated until both continuity and momentum equations are solved to an acceptable accuracy.

The problem identified in par. 7.9.6 is addressed as follows: In equation (7.94) the density at time step \( n+1 \) was present. A possibility is the computation of the density by solving the transport equations using the temporal velocity field \( u^* \) and \( w^* \). So the first term of the NS equation is written as:

\[ \frac{\partial}{\partial t} \int_{\Omega} \rho u \, d\Omega = \frac{(\rho u^*) - (\rho u)^n}{\Delta t} \Delta x \Delta z \] (7.103)

A drawback of this approach is that this method has a tendency to unstable behaviour. Therefore the term is simplified to the following form:

\[ \frac{\partial}{\partial t} \int_{\Omega} \rho u \, d\Omega = \frac{(\rho u^{n+1}) - (\rho u)^n}{\Delta t} \Delta x \Delta z \] (7.104)

160
Hence the density of the last time step is used. It was investigated if the solutions would be different if a prediction of $\rho^{n+1}$ was based on the flowfield at time $n$. The procedure was as follows:

Use the flowfield $u^n$ and $w^n$ to update an approximated density for the next time step denoted with $\tilde{\rho}^{n+1}$. Use this density to compute:

$$\frac{\partial}{\partial t} \int_{\Omega} \rho u \, d\Omega = \left( \frac{\tilde{\rho}^{n+1} u^{n+1}}{\Delta t} - (\rho u)^n \right) \Delta x \Delta z$$  \hspace{1cm} (7.105)

No differences between the results obtained with equation (7.104) and equation (7.105) could be observed, hence in the model the simple approach according to equation (7.104) was used.

### 7.10 Numerical implementation of the $k$-$\varepsilon$ model

In this Section the attention is focussed on the implementation of the $k$-equation. The $\varepsilon$ - equation is treated the same way. The $k$-equation reads:

$$\frac{\partial k}{\partial t} + \frac{\partial (uk)}{\partial x} + \frac{\partial (wk)}{\partial z} = \frac{\partial}{\partial x} \left( \nu \frac{\partial k}{\partial x} \right) + \frac{\partial}{\partial z} \left( \nu \frac{\partial k}{\partial z} \right) + P + P_e - \varepsilon$$ \hspace{1cm} (7.106)

The following issues regarding the numerical approximation of the turbulence equations are of importance:

- The generation of negative values of $k$ or $\varepsilon$ must be avoided at all times, otherwise a negative eddy-viscosity may arise. This would result in shear stresses pointing in the wrong direction leading to instability in the numerical procedure.
- The local balance of production and dissipation of turbulence is of primary importance when solving the $k$-$\varepsilon$ equations. Transport is therefore less important, so conservation and high precision of advection is not of great significance (Stelling (2000)).

In Figure 7.7 the double staggered grid arrangement was highlighted. Consider now the gridpoint where the eddy-viscosity, $k$ and $\varepsilon$ are defined (Point P in Figure 7.10). The gridpoints for the horizontal and vertical velocity are located at the wrong places for the application of a Finite Volume Method. For the FV method it is preferable to have the horizontal velocities on the west and east faces, and the vertical velocity on the north and south faces (and not the other way around like in Figure 7.10).
Figure 7.10 Staggered grid centred on a gridpoint for \( k \) and \( \varepsilon \).

In order to get the values at the face centres averaging between surrounding gridpoints cannot be avoided. Since conservation is less important solving the transport equations of the turbulent quantities the Finite Difference Method is used. For the advection an upwind differential scheme is used to ensure positive solutions. A higher order scheme does not lead to better results as shown by Zijlema (1994). The different terms in the transport equation will now be derived.

### Unsteady term

A first order (Euler) approximation is used:

\[
\frac{\partial k}{\partial t} = \frac{k_{p}^{n+1} - k_{p}^{n}}{\Delta t}
\]  

(7.107)

### Advection

An Upwind Differential Scheme is used for this term. The horizontal velocity in point P is found by linear interpolation between the values at gridpoints n and s. The following, always positive, expression is found:

\[
\frac{\partial}{\partial x} (u_k) = u \frac{\partial k}{\partial x} = \begin{cases} 
\frac{u_{p}^{n+1} k_{p}^{n+1} - k_{n}^{n+1}}{\Delta x_{n-1}} & u_{p}^{n+1} > 0 \\
\frac{u_{p}^{n+1} k_{p}^{n+1} - k_{s}^{n+1}}{\Delta x_{i}} & u_{p}^{n+1} \leq 0 
\end{cases} 
\]  

(7.108)

This formulation is implicit for \( k \). For the velocity the flowfield for the new time step is used. This is possible because in the computation the first step is updating the momentum equations using the viscosity of the previous time step.

The advection in vertical direction is treated correspondingly.

162
Diffusion

The horizontal diffusion is approximated as follows:

\[
\frac{\partial}{\partial x} \left( \frac{v_e}{\sigma_k} \frac{\partial k}{\partial x} \right) = \frac{1}{\sigma_k} \frac{v_e}{\sigma_k} \frac{\partial k}{\partial x} \left( \frac{v_e}{\partial x} \right) + \frac{1}{\sigma_k} \frac{v_w}{\partial z} \left( \frac{v_w}{\partial z} \right) \left( \frac{\partial k}{\partial z} \right) \frac{1}{\Delta x + \Delta x_{-1}}
\]  

(7.109)

The terms at the "e" and "w" (see Figure 7.10) locations contain the eddy-viscosity and the gradient of k. These quantities are computed by averaging and central differences respectively:

\[
\left( \frac{v_e}{\partial k} \right) \approx \frac{1}{2\sigma_k} \left( (v_e^e)^2 + (v_e^w)^2 \right) \frac{k_{e+1} - k_{e+1}}{\Delta x_{-1}}
\]  

(7.110)

Again the formulation is implicit. For the eddy viscosity the information of the previous time step is used. The vertical diffusion terms are treated likewise.

Production \( P \)

The production term reads (equation (7.33)):

\[
P = 2\nu \left[ \left( \frac{\partial u}{\partial x} \right)^2 + \frac{1}{2} \left( \frac{\partial u}{\partial z} + \frac{\partial v}{\partial x} \right)^2 + \left( \frac{\partial w}{\partial z} \right)^2 \right]
\]  

(7.111)

The central term in this expression has the largest influence since this represents the shearing of the fluid, which gives the largest contribution to the generation of turbulence. Because of the advantage of using the double staggered grid the gridpoint for \( k \) is nicely situated between the necessary gridpoints for the horizontal and vertical velocity. The horizontal velocity at the north and south for the horizontal shearing and the vertical velocity at the east and west side for the other direction. No averaging or interpolation is needed which contributes clearly to the accuracy. These terms are approximated as follows:

\[
\frac{\partial u}{\partial z} = \frac{u_{n+1} - u_n}{\frac{1}{2}(\Delta z_k + \Delta z_{k-1})} \quad \frac{\partial w}{\partial x} = \frac{w_{n+1} - w_n}{\frac{1}{2}(\Delta x_i + \Delta x_{-i})}
\]  

(7.112)

For the approximation of the other two gradients interpolation or averaging is needed. For instance for the term \( \partial u / \partial x \) the horizontal velocity at location east and west is required. These contributions are computed by averaging the velocities of the four grid points surrounding the east and west location respectively. Since all gradients are squared the production term remains positive at all times and no additional measures have to be taken.
Dissipation $\varepsilon$

The dissipation $\varepsilon$ needs some extra attention since this term is negative in the $k$-equation which can possibly lead to negative values of the turbulent energy, which must be avoided to ensure stability. The first method to ensure positive values is standard Newton linearization (Zijlema 1993). In general can be written for a term $S$:

$$S_{n+1}^n = S_n^n + \frac{\partial S_n^n}{\partial \phi}(\phi^{n+1} - \phi^n) \quad (7.113)$$

Application of this method on the dissipation term using the general expression for the eddy-viscosity leads to:

$$-\varepsilon^{n+1} = -c_\mu \frac{(k^2)^n}{\nu_e^n} = -\left(c_\mu \frac{(k^2)^n}{\nu_e^n} + c_\mu \frac{2k^n}{\nu_e^n}(k^{n+1} - k^n) \right) =$$

$$= c_\mu \frac{(k^2)^n}{\nu_e^n} + 2c_\mu \frac{k^n}{\nu_e^n} k^{n+1} = \varepsilon^n - 2 \frac{\varepsilon^n}{k^n} k^{n+1} \quad (7.114)$$

The negative term is now decomposed into two terms. A positive explicit term in the right hand side and a positive matrix coefficient (implicit term).

Another method is transformation of the dissipation to an implicit contribution by multiplication and division by $k$ (Stelling, 2000):

$$-\varepsilon^n = -\varepsilon^n \frac{k^{n+1}}{k^n} \quad (7.115)$$

Again a positive matrix coefficient ($\varepsilon^n / k^n$) is obtained.

Buoyancy Production $P_b$

The buoyancy term in the $k$-equation reads:

$$P_b = \frac{g v_e}{\rho} \frac{\partial \rho}{\partial z} \quad (7.116)$$

The production due to buoyancy can be positive or negative depending of the sign of the density gradient (stable or unstable stratification). To maintain positive solutions the same method as used for the dissipation term is used. The term is treated explicit when positive and implicit when negative:

$$P_b = \begin{cases} 
\frac{g v_e}{\rho} \frac{\left(\rho_n^e - \rho_s^e\right)}{\rho_n^e \sigma \frac{1}{2}(\Delta z_k + \Delta z_{k-1})} & \rho_n^s - \rho_s^e \geq 0 \\
\frac{g v_e}{\rho} \frac{\left(\rho_n^e - \rho_s^e\right)}{\rho_n^e \sigma \frac{1}{2}(\Delta z_k + \Delta z_{k-1})} \frac{k^{n+1}}{k^n} & \rho_n^s - \rho_s^e < 0
\end{cases} \quad (7.117)$$
7.11 Treatment of the Boundaries

In the previous paragraphs the numerical approach for the interior cells was described. Due to the moving bed and water surface the top and bottom cells need a special treatment, which will be discussed in this section.

Implementation of the moving bed

As discussed in par. 7.7.2 a Cartesian grid with the sandbed moving through the grid was applied. Suppose that at a certain time the location of the bed is situated as indicated in Figure 7.11 with the dotted line. The bed boundary of the lowest grid cell intersects with the bed at the (horizontal) cell centre. In the figure two control volumes are indicated. The control volume on the left is for the transport of sediment. The control volume at the right is for the horizontal velocity.

Figure 7.11  Staggered grid arrangement near the sand bed.

In general the sand bed is not horizontal. Therefore cell heights of adjacent cells differ. This difference must be taken into account when calculating fluxes across the cell boundaries to preserve conservation. Here always the smaller of the two adjacent cells is used. The approximation of the advection term of the sediment transport equation can be written more generally as:

$$\int e \vec{v}_z \cdot \vec{n} \ dS = -\Delta z_n u_n c_n + \Delta z_e u_e c_e - \Delta x w_c c_c + \Delta x w_c c_n$$  \hspace{1cm} (7.118)

The length of the north and south faces are always equal since all gridlines are arranged vertically. The length of the west and east faces are computed as follows:

$$\left(\Delta z_n\right)_i = \min\left(\Delta z_i, \Delta z_{i-1}\right)$$

$$\left(\Delta z_e\right)_i = \min\left(\Delta z_i, \Delta z_{i+1}\right)$$  \hspace{1cm} (7.119)
Although other methods are possible, the most important issue whatever method used, is that it must be applied consequently in all grids (sediment transport, horizontal momentum, vertical momentum and the pressure correction method).

The new location of the sand bed is calculated every time step. The vertical velocity of the bed follows from equation (7.73). After some time a bed cell is totally filled and the bed grid cell is moved one place upward. The amount of sediment moving from the flow into the bed must be equal to the quantity stored in a “filled cell” (the dotted cells in Figure 7.11). This is achieved as follows. The total volume of a fraction stored in a bottom cell during filling is:

$$S_i = \sum_{j=1}^{k} c_i(t_j) \left(-v_{z,j}(t_j) + v_{sed}(t_j)\right) \Delta t$$  \hspace{1cm} (7.120)

Where $k$ the number of time steps that was needed to fill the bottom cell. The concentration in the bed of that fraction follows simply from:

$$c_{i,bed} = \frac{S_i}{\Delta z}$$  \hspace{1cm} (7.121)

The value of $S_i$ must remain positive; otherwise negative bed concentrations are computed, which is physically impossible. A negative value would occur when the vertical velocity of a fraction would be positive (in upward direction, which is not unrealistic) and larger than the sedimentation velocity. Using equation (7.73) the sedimentation velocity can be expressed as:

$$v_{sed} = \frac{1-c_i}{1-n_b-c_i} v_w$$  \hspace{1cm} (7.122)

Since the total concentration is bounded between 0 and $(1-n_b)$, $v_{sed}$ is larger than $v_w$. It is assumed that all grains slip in vertical direction relative to the water velocity (equation (7.43)), therefore the sedimentation velocity is always higher than the grain velocity and the condition $S_j > 0$ is assured. Bed concentration remains positive in all circumstances. The sedimentation velocity is reduced with a reduction factor $R$ depending on the value of the bottom shear stress. The reduction factor is computed using equation (7.74). The product of this reduction factor and $S_j$ is prescribed as the sediment flux leaving the bed-cell.

**Cell Merging**

During sedimentation the height of the bed cell decreases with time. Numerical problems (slow convergence or instability) occur when the height of a cell drops below a critical value. To avoid these problems cell merging is used (Yang et al. (1999)). When the cell height gets smaller than a factor times the original height, the location of the bed cell is moved one place upward. The initial height of that cell is increased with 166
the remains of the old bed cell. The concentration in the new bed cell is a weighted average between both cells that are merged to keep the method conservative. The height of the new and old bed cells is \( \Delta z_k \) and \( \Delta z_{k-1} \) respectively. The concentration in these cells before merging was \( c_{i,k} \) and \( c_{i,k-1} \). After merging the height and concentration in the bed cell follows from:

\[
\Delta z_{k}^{\text{new}} = \frac{\Delta z_k + \Delta z_{k-1}}{2},
\]

\[
c_{i,k}^{\text{new}} = \frac{c_{i,k} \Delta z_k + c_{i,k-1} \Delta z_{k-1}}{\Delta z_k + \Delta z_{k-1}}.
\] (7.123)

The control volume for the horizontal velocity is indicated in Figure 7.11. For the flux terms the west and east face height is \( \Delta z_{u,w} \) and \( \Delta z_{u,e} \) respectively. For the volume terms in the momentum equation the average height \( \Delta z_{u,p} = (\Delta z_{u,w} + \Delta z_{u,e}) / 2 \) is used.

**Pressure term at the bed**

It seems logical to apply the pressure on the actual height of the west \( \Delta z_{u,w} \) and east \( \Delta z_{u,e} \) face of the control volume. By doing so the forces due to hydrostatic pressure on the west and east face are unbalanced. In reality this resultant force is balanced with the horizontal component of the pressure acting on the sloping bed. No provision is available in the scheme to include such a force. This could of course be facilitated for the bed cells, but instead another approach is followed. For the pressure term the average cell height \( \Delta z_{u,p} \) is used. In that case an imbalance is still present since the vertical position of the pressure points at the east and west face differ slightly. This resultant force is equal to:

\[
\Delta F_k = \Delta z_{u,p} g \left( \rho_{\text{west}} \bar{z}_w - \rho_{\text{east}} \bar{z}_e \right)
\] (7.124)

This term is used in the horizontal momentum equation as an explicit correction term and no artificial flow near a sloping bed develops.

**Implementation of the movable surface**

The situation at the water surface is comparable to the bed. Depending on the level of the water surface relative to the grid points, the height of the top cells is adjusted. Due to the rigid lid assumption the water surface is horizontal and all top cells have the same height. Cell merging is used when needed.
7.12 Application of the 2DV Model

7.12.1 Introduction

Before the above described numerical model is applied on the hopper sedimentation process the model is tested on some less complicated flow problems to check if the numerical implementation of the equations has been performed correctly.

7.12.2 Channel flow clear water

The clear water flow in a rectangular channel is a simple flow problem where both the distribution of velocity and eddy viscosity are well known. The velocity profile can be calculated using (see (7.54):

\[ \frac{u(z)}{u_*} = \frac{1}{k^*} \ln \left( \frac{32z}{k_*} \right) \]  

(7.125)

With \( u_* \) as the friction velocity defined with (7.55). In open channel hydraulics the friction velocity can be directly related to the water depth \( h \) and the bed slope \( i \) in stationary conditions:

\[ u_* = \sqrt{ghi} \]  

(7.126)

At the free water surface a rigid lid approach is used, the hydraulic gradient can be related to the dynamic pressure gradient or the friction factor can be used to relate the friction velocity to the average velocity. Another method is to relate the friction velocity with the depth averaged velocity using:

\[ u_*^2 = \frac{f}{8} \bar{u}^2 \]  

(7.127)

The Darcy-Weisbach friction coefficient is calculated with:

\[ f = \frac{8g}{(18 \log (12R/k_*))^2} \]  

(7.128)

The logarithmic velocity distribution combined with a linear shear stress distribution leads to the following parabolic distribution for the eddy-viscosity as a function of the non-dimensional height \( z/h \):

\[ \nu_* = \nu u_* h \left( \frac{z}{h} \right)^2 \]  

(7.129)

The velocity profile is calculated using the standard and RNG \( k-\varepsilon \) model. The bed roughness is set to 0.5 mm. The shear stress at the surface is set to zero. The numerical results are compared with the logarithmic velocity profile and parabolic eddy-viscosity distribution in Figure 7.12. The difference between the computed velocity profile and
equation (7.125) is small. The influence of the type of turbulence model on the velocity distribution can be neglected. When the distribution of the eddy-viscosity is compared it becomes clear that the computed eddy viscosity is somewhat lower than according to equation (7.129) in the figure results of the same computation by Bijvelds (2001) are plotted. The results of Bijvelds are most closely reproduced with the RNG- $k-\varepsilon$ model.

![Graph showing velocity and eddy-viscosity profile for flow in rectangular channel.](image)

**Figure 7.12** Velocity and eddy-viscosity profile for flow in rectangular channel.

### 7.12.3 Backward facing step

The flow over a backward-facing step (a sudden expansion of the flow geometry) is a popular benchmark for testing CFD software. Measurements of Makiola, 1992 are used to compare the model results. Experiments were carried out in a closed duct with Reynolds numbers, based on the velocity and height at the inlet zone, ranging from 5000 - 64000. Three different expansion ratios were tested. This ratio is defined as flow height $h_1$ before and after the step $h_2$. The backward-facing slope of the step was varied as well between 10 - 90 degrees.

The following experiment was simulated:

- **Reynolds Number:** $Re = u_m h_1 / \nu = 15000$
- **Expansion Ratio:** $E_r = h_2 / h_1 = 1.48$
- **Angle:** $90^\circ$ (vertical step)

Both the standard- and the RNG $k-\varepsilon$ model were used. The flow in a closed duct is simulated, so instead of a free water surface a wall with the same roughness as the bed was applied at the top. The wall roughness was not mentioned in the experimental results (hydraulic smooth?), it appeared however that using a value of $k_c = 1$ mm reasonably reproduced the velocity profile at the inflow zone. In Figure 7.13 the numerical model is compared with the experiment. (only the results for the RNG $k-\varepsilon$}
model are shown). The velocity profile is reproduced very well. The reattachment length is underestimated by the numerical method. For this experiment the reattachment length was \(L/H = 6.3\).

![Graph showing simulation of turbulent flow over a backward-facing step.](image)

**Figure 7.13 Simulation of turbulent flow over a backward-facing step.**

The results for the different turbulence models are very small. The RNG-model predicts a slightly higher reattachment length: \(L/H = 6.0\) instead of 5.0 with the standard \(k-\varepsilon\) model. It is known that the \(k-\varepsilon\) model underpredicts the length of this zone by 10-20\%. It can therefore be concluded that this test case is well reproduced by the model.

Makiola (1992) also published experimental results with inclined steps. The following test was chosen for comparison:

- Reynolds Number: \(Re = u_m h_1 / v = 64000\)
- Expansion Ratio: \(E_r = h_2 / h_1 = 2\)
- Angle: \(25^0\)

In Figure 7.14 the experiments are compared with the numerical results. The measured reattachment length is 7.3, which is close to the calculated value. The calculated circulation velocity in the wake behind the step is however lower than measured.
Figure 7.14  Simulation of turbulent flow over a 25° backward-facing step.

7.12.4 Buoyant and non-buoyant plane jets

In the area where the flow enters the hopper the sand-water mixture flows towards the bed as a plane buoyant jet. It is important that this process is described adequately by the model. It is investigated if the buoyant and non-buoyant flow field, resulting from a jet entering in a basin, is reproduced with the model.

Non-buoyant plane jet

The spreading rate and development of velocity in plane jets in non-buoyant situations have been studied for a long time. The experimental results can be compared with the numerical predictions. Very often the width of the jet is defined as the point where the axial velocity is half the maximum velocity present in the centre of the jet.

Figure 7.15  Flow from plane nozzle.

This half-width $b(z)$ is expressed as:

$$\frac{b(z)}{D} = K_1 \left( \frac{z}{D} + K_2 \right)$$

(7.130)
Where $D$ is the width of the (plane) nozzle and $z$ the distance from the nozzle in flow direction. The spreading angle, defined by $K_s$, is often assumed to be constant and varies in different publications between 0.09 and 0.13. Kotsovinos (1976) compared several experimental investigations and reported that the spreading rate was not constant, but increased with the distance from the slot. It appeared that all results fitted quite well with the following relationship:

$$\frac{b(z)}{D} = 0.288 + 0.0913\left(\frac{z}{D}\right) + 5.101 \cdot 10^{-5}\left(\frac{z}{D}\right)^2 + 3.31 \cdot 10^{-7}\left(\frac{z}{D}\right)^3$$  \hspace{1cm} (7.131)

The velocity profile of the jet resembles a normal distribution at larger distance from the nozzle. The centreline velocity $u_m$ decreases with distance from the nozzle. The relation between the centreline velocity in a plane jet with distance is generally written as Chen & Rodi (1980):

$$\frac{u_m}{u_0} = A_u \left(\frac{z}{D}\right)^{-1/2}$$  \hspace{1cm} (7.132)

Where $u_0$ is the exit velocity in the slot. For $A_u$ Chen and Rodi (1980) recommended a value of 2.40. These two relations are used to validate the model. For the computations a grid was used as plotted in Figure 7.16. To improve numerical accuracy the grid is compressed near the jet centre and the bed.

![Figure 7.16 Finite Volume Grid for jet computations.](image)

In the left panel of Figure 7.17 the location of the nozzle is located at the origin. The fluid exits the slot in vertical direction. The computed vertical velocity profiles are plotted for different distance from the exit point. The normal distribution can be seen clearly together with the decay of the velocity as function of the distance from the slot. The symbols indicate the position where the vertical velocity is half the centreline velocity (the half-width of the jet). The curved continuous line drawn from the origin is
the half-width value computed with the empirical relation (7.131). The value of the centre line velocity normalised with the exit velocity is shown in the right panel of the figure. The computed value is compared with the empirical relation (7.132). The standard \( k - \varepsilon \) turbulence model was used with the standard set of coefficients. It is clear that the predicted spreading rate is too small, which is a familiar behaviour of this turbulence model for these type of flows (Rodi (1982). The narrower spreading of the jet is accompanied with an overestimated centre line velocity.

**Figure 7.17** Development of plane non-buoyant jet (standard \( k - \varepsilon \) model).

The same calculation is repeated using the RNG-\( k - \varepsilon \) model. The results are shown in Figure 7.18. The spreading of the jet is now more in agreement with the empirical information, except for large distance where the upper boundary is influencing the results. The centreline velocity is as with the standard model somewhat larger than the value following from equation (7.132).

**Figure 7.18** Development of plane non-buoyant jet (RNG-\( k - \varepsilon \) model).
Buoyant Jets

The development of a plane buoyant jet exiting a plane nozzle is investigated below. The density at the nozzle is lower than the surrounding fluid. The spreading rate of a buoyant jet is larger than a non-buoyant jet (coefficient $K_1 = 0.116$, Chen & Rodi, 1980). Relatively close to the nozzle the momentum prevails over buoyancy destruction and the development of the centreline velocity decays as in a pure jet: $u::\sqrt{D/x}$ (see equation (7.132)). At larger distance the centreline velocity becomes constant and can be computed with:

$$\frac{u_m}{u_0} = B_u F^{-1/3} \left( \frac{\rho_0}{\rho} \right)^{1/3}$$  \hspace{1cm} (7.133)

Where $F$ is the densimetric Froude number at the nozzle:

$$F = \frac{u_0^2}{gD(\rho - \rho_0) / \rho_0}$$  \hspace{1cm} (7.134)

For the decay coefficient $B_u$ a value of 1.98 is recommended. The density of the surrounding fluid is $\rho$. Density of the discharged fluid is $\rho_0$.

In Figure 7.19 the calculated half-width is plotted versus the distance from the nozzle. The empirical half-width using $K_1 = 0.116$ (Rodi) is shown as well in this figure. The thin continuous line indicates the result of the $k-\varepsilon$ model with standard coefficient $c_\mu = 0.09$. It is shown that the calculated spreading of the jet is underpredicted.

![Figure 7.19 Calculated development of the width of a buoyant jet.](image)

This deficiency of the turbulence model for these buoyant flows has been reported in the literature. According to Yan & Holmsted (1999) and Shabbir & Taulbee (1990), the main problem is the description of the buoyancy term in the $k$-equation with a simple gradient type expression (equation (7.34)). Consequently the magnitude of the

174
production of turbulent energy due to buoyancy (in an unstable situation) is too small and hence the eddy-viscosity becomes too low. By using a more complicated turbulence model (Reynolds Stress Model) the numerical results improved significantly. Nam & Bill (1993) followed a simpler approach. They varied the model coefficient $c_\mu$ to increase viscosity and arrived at an optimal value of $c_\mu = 0.18$. This approach is used here. Two calculations were executed with the same value of this coefficient and the results are plotted in Figure 7.19. With this altered value the spreading of the jet is more in agreement with the experimental results. This can also be shown using the centre line velocity in the jet. Rodi (1982) presented the velocity as a function of the distance in non-dimensional terms:

$$B_m = F^{1/3} \left( \frac{\rho_m}{\rho} \right)^{1/3} \frac{u}{u_0} = f \left( F^{-2/3} \left( \frac{\rho_m}{\rho} \right) \frac{x}{D} \right)$$

(7.135)

The calculated velocity in the centre of the jet is now compared with measurement collected by Rodi (1982) in Figure 7.20. This figure shows that at larger distance buoyancy dominates: the jet acts as a buoyant plume. The recommended value for $B_m$ is 2, which is close to the model results for an increased value for $c_\mu$. The velocity is exceeded by approximately 25% for the standard values for the coefficients. The measurements show even lower values.

![Figure 7.20 Development of centreline velocity in a buoyant jet.](image)

Summarising the results of the model so far, it can be concluded:

- The 2DV model with the $k - \varepsilon$ turbulence model shows the expected behaviour.
- This behaviour implies however that the vertical velocity of the buoyancy driven flow in the inflow section of the hopper may be too large when the standard values of the turbulence model are used.
7.12.5 Sediment transport in a rectangular channel

So far all calculations were based on homogeneous fluids or non-settling mixtures where buoyancy was incorporated. Now the behaviour of the model for settling mixtures in free surface channel flow is investigated. Relatively low concentrated suspended sediment transport is investigated first. The average concentration is 2% by volume, water depth \( h \) is approximately 0.1 m and the specific discharge is 0.15 m\(^2\)s\(^{-1}\). The friction velocity \( u_\ast = 0.09 \) m/s and the \( d_{50} \) of the particle size distribution is 120 \( \mu \)m. The grain size distribution is schematised by using 8 fractions. For low concentrated suspended sediment transport the concentration distribution as a function of depth can be described with the Rouse distribution:

\[
\frac{c}{c_a} = \left( \frac{z_a}{h-z_a} \right)^Z
\]

(7.136)

The exponent in this expression \( Z \) is the suspension parameter defined as:

\[
Z = \frac{w_f}{\beta x u_*}
\]

(7.137)

The \( \beta \) coefficient is defined with equation (7.42) and \( c_a \) is the reference concentration near the bed at level \( z_a \). The Rouse distribution can be determined when the values of \( z_a \) and \( c_a \) are known. For \( z_a \) the vertical position of the centre of gridcell at the wall is chosen. For \( c_a \) the value of the concentration computed with the 2DV model in this cell. The model reproduces the Rouse distribution quite accurately, as can be seen in Figure 7.21 where the computations are compared with the Rouse distribution.

![Figure 7.21](image)

*Figure 7.21  Velocity and concentration profile at low concentration.*
7.12.6 Hyperconcentrated flow

Concentrations in the hopper can be as high as 0.2 – 0.4. These types of suspensions are called hyperconcentrated suspensions. High concentration affects the flow in several ways. The particles are kept in suspension by turbulence over the largest part of the flow depth. Grain-grain interactions contribute to the suspension of sediment close to the bed. The settling flux is balanced by turbulent suspension accompanied with a concentration gradient. This gradient stabilises the flow and damps turbulence (see Section 7.3.5).

The presence of the grains increases the effective viscosity of the mixture. This leads to increased damping of turbulence due to viscous dissipation. Winterwerp et al. (1990) performed experiments in a so-called tilting flume. Highly concentrated sand-water mixtures are kept in suspension in free surface flow in a relative steep sloping flume. From the original report of Mastbergen et. al. (1987) some of the experimental results were extracted and simulated using the present 2DV model. A large number of tests with two sands and different flowrates and mean concentration $\bar{c}$ (up to 40%) were performed in the test programme. Four tests were chosen arbitrarily and compared with the results of the 2DV model.

**Test conditions**

The flow was supercritical with a specific flow rate of 0.15 m$^3$/s and median grain diameter $D_{50} = 120$ $\mu$m. On the bottom of the flume coarse sand was glued with a median grain $D_{50}$ of 500 $\mu$m. Table 7.3 shows the test conditions of the selected experiments. During the all tests a stationary bed was absent – all particles were in suspension.

<table>
<thead>
<tr>
<th>test</th>
<th>$\bar{c}$ [%]</th>
<th>depth [m]</th>
<th>$i_{flume}$ [-]</th>
<th>$i_{eq}$ [-]</th>
<th>$u_*$ [m/s]</th>
<th>$\theta$ [-]</th>
<th>temp [°C]</th>
</tr>
</thead>
<tbody>
<tr>
<td>57</td>
<td>10.8</td>
<td>0.0981</td>
<td>0.0102</td>
<td>0.0102</td>
<td>0.099</td>
<td>5.0</td>
<td>24.9</td>
</tr>
<tr>
<td>51</td>
<td>20.9</td>
<td>0.1081</td>
<td>0.0071</td>
<td>0.0072</td>
<td>0.087</td>
<td>3.9</td>
<td>25.9</td>
</tr>
<tr>
<td>46</td>
<td>31.9</td>
<td>0.1166</td>
<td>0.0059</td>
<td>0.00415</td>
<td>0.076</td>
<td>3.0</td>
<td>26.7</td>
</tr>
<tr>
<td>43</td>
<td>42.2</td>
<td>0.1102</td>
<td>0.0066</td>
<td>0.0059</td>
<td>0.084</td>
<td>3.6</td>
<td>30.6</td>
</tr>
</tbody>
</table>

*Table 7.3 Overview selected test of the Tilting Flume Experiments.*

Columns 3-5 need extra explanation. The third column provides the flow depth during the experiment. The fourth column gives the slope of the flume during the test. When the slope of the flume is steep enough, no sedimentation takes place in the flume. When at a certain specific flowrate and concentration the slope is decreased gradually, sedimentation starts to occur at a certain equilibrium slope $i_{eq}$. The value of $i_{eq}$ is shown in column 5.
Boundary conditions used for the computations.
The bed roughness was taken equal to \( k = 2D_{s0} \). During tests 57 and 51 the slope angle was close to the equilibrium value. This can also be seen from the value of the Shields parameter \( \theta \) in the Table, which is close to the threshold value \( \theta_0 = 4 - 5 \) observed in the closed flume sedimentation tests described in Chapter 5. The sedimentation velocity at the bed in the 2DV model is modelled using equation (7.75). During the computations the value of the reduction factor was zero (\( R(\theta) = 0 \)). The expressions of equation (7.65) are used for the turbulent quantities at the bed. At the water surface, \( k = 0 \) and for \( \varepsilon \) boundary condition (7.64) was used. In Figure 7.22 the experiment is compared for a concentration of 10%. The left panel shows the velocity profile and the right figure the concentration profile. The results of four different computations are plotted in this figure beside the experimental results (symbols). The particle size distribution was approximated using 6 different fractions (multi-sized mixture) for three computations. One calculation was based on a mono-sized sediment (all grains have the same diameter of 120 \( \mu \text{m} \)). Beside the Particle Size Distribution the bottom roughness height \( k \) of the flume was varied between 0.5 and 1 mm.

\[
\begin{align*}
\text{Figure 7.22} & \quad \text{Velocity and concentration profile for } c=10\% \text{ (exp. 57).}
\end{align*}
\]

When the computations are compared with the experiments for \( c = 10 \% \), the following observations can be made:

- The computed velocity profile is less curved than the measured profile. Especially for roughness height \( k = 1 \text{ mm} \) and \( k = 1.5 \text{ mm} \) the velocity profile is a straight line from \( z/h = 0.3 - 1 \).
- The computed concentration profile based on one fraction shows the largest difference with the measurements.
The computation based on \( k = 0.5 \text{ mm} \) yields the best near bed concentration. The velocity distribution for this case shows however the largest difference with the measurements.

The computed linear velocity distribution needs explanation, since this type of profile usually occurs when grain collisions become important. The latter effect is not to be expected with this fine sand. The velocity profiles are investigated below. The shear stress \( \tau \) at distance \( z \) from the bed follows from:

\[
\frac{\partial \tau}{\partial z} = -\rho(z) gi
\]  
(7.138)

Where \( i \) is the slope of the flume. If the density of the mixture is uniform over depth this leads to a linear shear stress distribution over depth, with the maximum value at the bed and zero at the water surface. If the density varies over depth (as in Figure 7.22) the values at the bed and surface remain the same, but in between the distribution is no longer linear, but quadratic. It can however be shown that for the situation of Figure 7.22 the deviation of the linear distribution is very small and therefore the shear stress can be regarded as proportional with distance \( z \):

\[
\tau = -\bar{\rho} g i (h - z)
\]  
(7.139)

The relationship between the shear stress and the velocity gradient in turbulent flow is:

\[
\tau = \rho \nu_e \frac{\partial u}{\partial z}
\]  
(7.140)

The linear shear stress distribution can only be achieved with equation (7.140) where the dynamic eddy-viscosity (\( \rho \nu_e \)) is linear with depth since the velocity gradient is constant over a large distance.

In Figure 7.23 the computed eddy-viscosity is shown for the profiles of Figure 7.22 (only the multi-sized results). The computed eddy-viscosity for clear water is also shown for reference. It is clear that owing to the presence of the concentration gradient turbulence is damped compared with the clear water situation. The expected linear decrease of the dynamic viscosity (needed to explain the constant shear rate) is shown clearly as well.
Figure 7.23  Computed eddy viscosity for c=10% and c=0% for different values of roughness height k.

The computations performed on higher concentrations are compared with the experiments in Figures 7.24 - 7.26. Beside the standard $k-\varepsilon$ model (in Section 7.3.5) two other variations of the model were used: The standard model with added 'background turbulence'; and the standard model without the buoyancy terms.

Figure 7.24  Velocity and concentration profile for c=20%. 

180
Figure 7.25  Velocity and concentration profile tilting flume experiments (c=30%).

Figure 7.26  Velocity and concentration profile tilting flume experiments (c=40%).

The following observations can be made:

Accuracy of Measurements
The 2DV model is not always responsible for the observed difference between the measured and computed profiles. Although the shape of the computed profiles may not be correct, continuity (both for total volume as for sediment) is always satisfied. The integrated measured velocity and concentration profile is sometimes not equal to the values reported in Table 7.3, see for instance the concentration profile of Figure 7.25. The difference can be caused by a non-uniform flow distribution over the width of the flume and/or measurement inaccuracy.
Standard $k - \varepsilon$ model

The (computed) concentration shows a sharp decrease near the surface while the velocity features the opposite behaviour for 20% and 30% mean concentration. This phenomenon is caused by the buoyancy term in the turbulence model. Since the diffusion of sediment is directly coupled with the eddy viscosity and the latter approaches to zero near the surface a concentration gradient near the surface develops. This gradient damps the eddy viscosity and consequently the turbulent diffusion, so the gradient increases. The (strongly) reduced eddy-viscosity leads to a low shear resistance of the flow and hence high velocity near the fluid surface. Although this phenomenon can occur in reality, the model exaggerates the effect. In the experiments the eddy-viscosity does not reduce to zero because the sidewalls in the flume and surface waves generate turbulence. These effects are not included in the model. Another influence, not included in the model, is the effect of the grain-grain interactions near the bed.

Modified $k - \varepsilon$ model.

To avoid unrealistically low values for the eddy-viscosity “background-turbulence” is added. The eddy-viscosity is consequently always above a certain value (in this case $10^{-4} \text{m}^2\text{s}^{-1}$). With this simple addition the turbulence induced by surface waves and sidewalls is included and both the concentration and the velocity profiles are improved. The unrealistic behaviour near the surface disappears.

$k - \varepsilon$ model without the buoyancy term.

Without damping by concentration gradients a more homogeneous concentration profile develops. The velocity profile resembles the clear-water distribution. Using the turbulence model without the buoyancy term provides reasonable results for this high concentration. For a concentration of 20% (Figure 7.24) the standard turbulence model still exaggerates the drop in concentration near the surface, but the difference between the measurements and the computations becomes less. The modified model with added turbulence agrees quite well with the measurements.

7.12.7 Concluding remarks hyperconcentrated flow

In the previous section the 2DV model was applied on highly concentrated supercritical flows with fine sediment. The relevance of this comparison for the sedimentation process in a hopper is reviewed below. The agreement between experiments and computations was reasonable as long as the concentration gradient is not too large. If the concentration gradient is localised near the surface (for mean concentration above 10%) the results improve when ‘background turbulence’ is added. Too strong damping of turbulence by the $k - \varepsilon$ model most likely causes the difference between measurements and computations when large gradients are present in the flow.
A large concentration gradient near the surface is present during the sedimentation process in a hopper as well, but at the bed large gradients as shown in Figure 7.22 (30% difference over only 0.1 m) are not present. Differences between measured and computed near bed concentration is less than 10% for the tests with mean concentration above 10%. The effect of this difference on the sedimentation velocity is limited because the sedimentation velocity is not too sensitive to the value of the near bed concentration (see Figure 4.14). It is therefore expected that the 2DV model can be applied for the hopper sedimentation process.

### 7.12.8 Buoyant plume on a sloping bed

The last case tested was the simulation of the behaviour of a density current on a sloping bed (see Figure 7.27). This type of flow occurs in a hopper (sand-water mixture) as well as in estuaries (saline water). From measurements it is found that the front speed of a density current is constant in case the Reynolds number is large enough and the slope is larger than or equal to 5 degrees. (Britter and Linden, 1980). In those circumstances bed friction and entrainment of ambient fluid balance the acceleration.

![Schematic of density current on slope with front speed \( u_f \).](image)

**Figure 7.27  Schematic of density current on slope with front speed \( u_f \).**

Under these conditions the normalised front speed \( u_{f,\text{norm}} \) is given by:

\[
u_{f,\text{norm}} = \frac{u_f}{(g' \dot{q})^{1/3}} = 1.5 \pm 0.2
\]

(7.141)

The front speed \( u_f \) is normalised with the buoyancy flux. The discharge per unit width is \( \dot{q} \), and \( g' \) is defined as \( g \Delta \rho / \rho \).

The computed translation of the front of the density current is shown in Figure 7.28. The slope angle was 5 degrees. The initial density of the plume is 1083 kg/m\(^3\) (5% concentration of sediment). The density of the ambient fluid is 1000 kg/m\(^3\). The value of the discharge was 0.0343 m\(^2\)/s. The upper figure shows the 1017 kg/m\(^3\) (1%) contour line with 5 s interval. The front speed is computed from the difference in horizontal position of the front of these contours.
The computed resultant normalised front velocity is plotted versus time in the lower figure. The computed velocity is very close to the experimental value of 1.5 given with equation (7.141).

7.12.9 Concluding remarks test calculations

In the previous sections the 2DV model was tested against various flow problems to investigate if the model can reproduce known flow situations. These situations involved clear water (channel flow and flow over a so-called backward facing step and plane jets) and sediment-laden flow where buoyancy effects are important (low and hyperconcentrated channel flow, buoyant jets and density currents over a sloping bed). From these calculations it can be concluded that the 2DV model is capable of providing a fair to reasonable prediction for most cases. Important differences between measurements and computations can occur at locations where buoyancy effects are dominant and large concentration gradients are present (buoyant jets and hyperconcentrated flows). For the application of the model for hopper sedimentation, these discrepancies may be important at the following locations:

- In the inflow section the sediment-water mixture flows in vertical direction towards the bed as a buoyant jet. In Section 7.12.4 we have seen that in case a plane jet is driven by buoyancy the centreline velocity in the jet is overpredicted when the $k - \varepsilon$ model with standard values is used.
- A large concentration gradient can only be present near the water surface in a hopper. Whether or not this phenomenon can be reproduced correctly by the model is investigated in the next section.
7.13 Application of the 2DV model for hopper sedimentation

7.13.1 Introduction
In the following sections the 2DV model is tested for the situations for which it was developed: Sedimentation of Sand in a Hopper. The important parameters influencing the sedimentation process were summarised in the introduction of Chapter 6. It is clear that an almost unlimited number of different simulations can be made with the model. In this thesis the emphasis has been given to validation of the model using the results of the experiments and a limited number of prototype measurements. In Section 7.13.5 the numerical model is therefore compared with a selection of different experiments carried out at Delft Hydraulics (large scale hopper sedimentation tests, see Chapter 4). In Chapter 8 the model results are compared with prototype measurements taken on board the Boskalis Westminster Trailing Suction Hopper Dredger “Cornelia”.

7.13.2 General settings of the 2DV calculations
In this thesis several methods have been presented to compute the settling velocity, the coefficient of the hindered settling function and various parameters of the numerical model, boundary conditions etc. Unless indicated otherwise, all computations presented in this Chapter have been executed using the following settings:

\textit{Sediment properties}
\begin{itemize}
  \item Setting velocity $w_0$ was determined by direct (iterative) solving of the equilibrium equation (3.4).
  \item The shape factor $\psi$ of the grains was assumed to be 0.7.
  \item The value of the exponent of the hindered settling function model was according to the values of Garside et al. (1977), see equation (3.16).
  \item The (specific) density correction according to Selim et al. (1983) was not used (see equation (3.21), because the particle size distribution of the sediment was uniform. Appendix D shows that application of this method cannot be recommended for situations with a uniform grain size.
  \item The mutual influence of the different fractions was modelled according to equation (7.45).
\end{itemize}

\textit{Boundary Conditions}
\begin{itemize}
  \item The wall-function approach was used for the momentum- and turbulence equations, see Section 7.5.1.
  \item Bed roughness $k$ was equal to 2.5 $D_{50}$.
  \item The boundary conditions were described in Section 7.5.2. For $\varepsilon$ equation (7.64) was used. The value for $k$ was equal to zero at the surface.
\end{itemize}

185
- The standard $k-\varepsilon$ was used with standard values of the coefficients (Table 7.1), because the RNG-$k-\varepsilon$ model did not give different results for the hopper sedimentation simulations.
- Turbulence intensity $I_r$ in the inflow was equal to 0.05 (Equation (7.67))

### 7.13.3 Influence of the grid size

In the previous section the influence of the inflow velocity was analysed. In the inflow section large gradients of concentration and velocity are observed. Investigations were carried out to ascertain whether the results are sensitive to the gridsize of the model.

The same geometry as in Figure 7.31 was chosen. The inflow velocity was 0.75 m/s. Figure 7.29 shows a detail of the grid near the inflow section. The grid is refined over a distance of 0.15 m below the point where the mixture enters the hopper. Four different grids were tested. The coarsest grid had 50 cells in horizontal direction and 60 in vertical direction ($50 \times 60$). The refined section contained 10 cells in horizontal direction and the flow entered into four cells only. The number of cells in the finest grid were $100 \times 80$. The refined section in this grid contained 40 cells of which 16 cells were used as inflow cells. The inflow width was thus kept the same for all grids.

![Figure 7.29](image)

**Figure 7.29** Detail grid of the $70 \times 60$ grid at the inflow section.

The inflow concentration was 0.30 for all grids. The horizontal velocity and concentration profiles from the different calculations are compared in Figure 7.30 (at 20 s after the flow was started). Two horizontal locations are chosen: close to the inflow section ($x = 0.35$ m) and approximately in the centre of the hopper ($x = 6.35$ m). The concentration is only shown at $x = 6.35$ m to avoid many cluttered lines. Figure 7.30 shows that the concentration and velocity distribution is not very sensitive to the resolution as long as the grid is not too coarse.

186
7.13.4 Influence of the inflow velocity

The influence of the inflow velocity for a certain discharge and concentration has been investigated. The flow field and concentration distribution during the first phase of test 6 was computed. The average specific discharge during this test was 0.0448 m²/s. The volume concentration at the start of the test was approx. 0.3. The inflow velocity \( u_0 \) was varied for the different computations between 0.45 and 3.0 m/s by variation of the thickness of the jet \( d_{in} \) entering the hopper (Figure 7.3). The initial water level (relative to the flume bottom) was at 1.25 m. The height of the sand bed at the start of the test was 0.18 m, so initial flow depth was 1.08 m.

The computed horizontal velocity profiles, at time \( t = 70 \) s after the start of the test, are shown in Figure 7.31. The computed horizontal velocity is plotted at nine different locations as a function of the vertical distance from the bed. The lines are plotted for different values of the inflow velocity. The position of the inflow is at the top left corner of the figure (\( x = 0 \) m, \( z = 1.6 \) m). Inflow direction is vertical. The horizontal velocity is positive near the bed, which implies that the flow is from left to right, towards the outflow section. Above \( z = 0.6 \) m the horizontal velocity is negative; the flow is directed towards the inlet zone.

The negative flow velocity is caused by entrainment of water into the downward flowing jet at the inflow zone. Horizontal velocity at the bed increases with the inflow velocity. The entrainment also increases with increasing inflow velocity. The circulation induced by the inflowing mixture intensifies slightly with the inflow velocity, which can be explained with the increased momentum of the inflowing mixture. The influence of
the inflow velocity at a larger horizontal distance $x$ on the velocity near the bed is relative small.

![Diagram showing velocity distribution](image)

**Figure 7.31** Computed horizontal velocity distribution in hopper ($t=70$ s, test 06).

The concentration profiles are shown in Figure 7.32. Here also a limited influence of the inflow velocity on the profiles is observed. The measured concentration is also plotted (at $x = 3.6$ and 9 m) in the figure. The measured concentration profiles show less mixing (less uniform) in the vertical direction.

![Diagram showing concentration distribution](image)

**Figure 7.32** Computed and measured concentration distribution in hopper ($t=70$ s, test 06).

The relatively small influence of the inflow velocity on the computed velocity and concentration distribution can be explained by the dominant effect of buoyancy.
In Section 7.12.4 it was shown that in case a mixture behaves like a buoyant plume the velocity in the centre of the plume \( u_m \) becomes constant after some distance and can be computed with:

\[
\frac{u_m}{u_0} = B_u F^{-1/3} \left( \frac{\rho_0}{\rho} \right)^{1/3}
\] (7.142)

Where \( u_m \) is the centreline velocity in the plume (which can be compared with the maximum velocity near the bed in Figure 7.31), \( u_0 \) is the inflow velocity and \( F \) the densimetric Froude number at the inflow, defined with:

\[
F = \frac{u_0^2}{g d_m (\rho - \rho_0) / \rho_0}
\] (7.143)

For the decay coefficient \( B_u \) a value of 2 is recommended (Rodi, 1982). The density of the surrounding fluid is \( \rho \). Density of the discharged fluid is \( \rho_0 \). The thickness of the jet at the inflow point is \( d_m \). The jet thickness and inflow velocity can be eliminated from these equations using the specific discharge \( q_0 = u_0 d_m \). Equation (7.143) can therefore be written as:

\[
\frac{u_m}{q_0} = B_u \left( \frac{q_0^2}{g \frac{\rho - \rho_0}{\rho_0}} \right)^{-1/3}
\] (7.144)

The flow velocity \( u_m \) only depends on the discharge and relative density difference at the inflow point. Both parameters were not varied for the different calculations of Figure 7.31. The computed velocities are not completely independent of the inflow velocity, which implies that the mixture is not acting as a pure buoyant plume. The jet behaviour is still partly present.

7.13.5 Model hopper sedimentation tests

The 2DV model was compared with the model sedimentation tests discussed in Chapter 4. The same tests were selected as were used for the validation of the IDV model (test 05 and test 06). In addition, test 04, performed on relative coarse sand, was also used for comparison.

1. Simulation of test 06

This experiment was already used as a benchmark for the 1DV model because of the high overflow rate, which resulted in large overflow losses. The (time averaged) operation parameters of the model tests used for validation are summarised in Table 7.4. The measured inflow concentration and discharge as function of time was used as input for the model.

189
Table 7.4  Conditions model tests used for validation of the 2DV model.

Some variation of the particle size diameter in the inflow was observed, but during the calculations the PSD was kept constant and represented with 8 fractions. Hence for the cumulative distribution 9 points are needed. Table 7.5 shows the cumulative distribution based on the average value of all samples taken during the test:

<table>
<thead>
<tr>
<th>Test 04</th>
<th>Test 05, calc 1</th>
<th>Test 05, calc 2</th>
<th>Test 06</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>0 5</td>
<td>0 5</td>
<td>0 5</td>
</tr>
<tr>
<td>48</td>
<td>2.6 39</td>
<td>7.3 48</td>
<td>14.3 39</td>
</tr>
<tr>
<td>91</td>
<td>11.0 73</td>
<td>23.0 91</td>
<td>42.2 73</td>
</tr>
<tr>
<td>134</td>
<td>39.9 106</td>
<td>49.1 134</td>
<td>72.8 106</td>
</tr>
<tr>
<td>178</td>
<td>74.4 140</td>
<td>70.5 178</td>
<td>94.6 140</td>
</tr>
<tr>
<td>221</td>
<td>91.8 174</td>
<td>91.1 221</td>
<td>97.2 174</td>
</tr>
<tr>
<td>264</td>
<td>99.0 208</td>
<td>95.9 264</td>
<td>98.4 208</td>
</tr>
<tr>
<td>307</td>
<td>99.3 241</td>
<td>97.7 307</td>
<td>99.0 241</td>
</tr>
<tr>
<td>350</td>
<td>100 275</td>
<td>100 350</td>
<td>100 275</td>
</tr>
</tbody>
</table>

Table 7.5  Overview Particle Size Distribution used in the simulations.

Concentration profiles as function of the time

Figure 7.33 shows the computed concentration profiles at four different positions for 9 different times. The inflow is located at the upper left side, the outflow at the right side. The overflow level was kept constant during the test at 2.25 m. The location of the calculated sand bed is shown with the hatched lines. The computed scour hole below the inflow point is clearly visible. The first profile in time was taken before the overflow level was reached. The other profiles were taken during the overflow phase.
A fair resemblance is present between the computed and measured concentration profiles as is shown in Figure 7.34, where the measured and computed concentration profiles are compared for one location \((x = 6 \text{ m})\). In the figure the lines with the symbols indicate the measured profiles.

- A sharp interface between the suspension and sand bed is observed.
- The profiles are almost vertical over a large (vertical) distance
- After some time (about 1000 s) the concentration in suspension becomes constant.
The measured value of the concentration is however larger than calculated, which might be caused by:
- The larger vertical diffusion already observed in Figure 7.32.
- The flow created a scour hole during the first phase of an experiment in the bed already present in the hopper. The eroded volume of sand enlarges the concentration of the suspension. This phenomenon is not included in the numerical model. The influence of the bed shear stress is modelled as a reduction of the sedimentation velocity. Net erosion is not possible with this approach.
- The calculated overflow concentration was larger than measured during the first phase of the loading process. Therefore more sediment was removed from the suspension.

**Horizontal velocity profiles**
The calculated velocity profiles at different times are plotted at 4 locations (at $x = 0, 3, 6$ and $9$ m) in Figure 7.35. It is clear that due to the density current the flow is concentrated close to the bed. Above this density current a return flow, caused by entrainment in the inflow section, is present. During time the magnitude of the horizontal velocity decreases, as can be seen from the maximum values of the profiles.

![Figure 7.35 Computed velocity profiles (test 06).](image)

The computed velocities can also be compared with measured values. This procedure is however more complicated due to the limited number of probes. The velocity profile could not be measured directly using several probes at the same time at the same horizontal position. An indication of the profile can be obtained by transforming the registration as a function of time of a single probe to space, assuming a constant sedimentation and flow velocity. This procedure was outlined in Section 4.9.3. Here another approach has been chosen. The position of all probes during time was recorded.
Most probes were fixed at a certain horizontal location \((x, y)\) and were raised during the experiment to follow the sand bed and to prevent burying. When the position of a probe is known as function of time the computed horizontal velocity at the same location can be recorded during the simulation. Figure 7.36 shows a result of this procedure. The horizontal position of the probe is \(x = 4\) m. The vertical position varies from \(z = 0.45 - 2.25\) m.

From this figure it can be learned that for this test the computed velocity is higher, although the same decreasing tendency during time is visible. At the end of the filling process, the velocity increases due to free surface flow. This occurs at the same time for both measurements and the computation. Figure 7.36 shows that also negative velocities, caused by entrainment were measured.

\[\text{measured} \quad \text{computed}\]

\(\text{hor. velocity [m/s]}\)

\(\text{time [s]}\)

Figure 7.36  Computed and measured horizontal velocity during test 06 (EMS 5).

The order of magnitude of the measured and computed velocity is the same and the model is run with all standard values for the coefficients (for instance for the turbulence model). No calibration was used. The difference between the computed and measured horizontal flow velocity can be caused by:

- Too much entrainment computed in the inflow section, where the buoyant jet flows vertically towards the bed.
- The influence of air bubbles entrained by the plunging jet during the experiments.

The buoyancy of the sediment-water mixture is decreased by the presence of the air bubbles, which leads to a lower driving force and hence to lower velocities. Further investigations are however necessary to prove this possible mechanism.

**Computed Overflow concentration**

A very important quantity during the calculations is the overflow loss. The computed concentration (for all fractions) in the overflow is compared with the measured value in Figure 7.37.

The agreement between theory and experiments is satisfactory. The maximum computed outflow concentration is somewhat smaller than measured and the computed
losses during the first part of the overflow phase are higher than measured. The latter being caused by the larger diffusion during that time, as already discussed above.

![Graph showing concentration over time](image)

*Figure 7.37 Measured and computed outflow concentration (test 06).*

The 2DV model reproduces the phase with constant overflow concentration quite well. The lower computed overflow concentration agrees with the lower computed suspension concentration. The higher value of the computed outflow concentration in the first phase was balanced with the lower values at a later stage which can be made clear by plotting the cumulative overflow loss computed with equation (4.25), see Figure 7.38, lower panel.

![Graph showing volume balance](image)

*Figure 7.38 Volume balance during simulation of test 06.*

The agreement between measured and computed overflow concentration is much better than the agreement between measured and computed suspended concentration (Figure 7.34). This seems contradictory on first sight, but is caused by the relatively low dependency of the sedimentation velocity on the near bed concentration (as shown earlier in Figure 4.14). The filling time of the hopper and likewise overflow loss is
therefore approximately the same. The upper panel of Figure 7.38 shows the total volume of grains in the hopper (settled plus in suspension) computed using two methods. The first method is by integration of the total concentration over the hopper area:

\[ V_1 = \int_{\Omega} c \, d\Omega \]  \hspace{1cm} (7.145)

The second method is by the difference between the total volume of in- and out-flowing sand:

\[ V_2 = \int_0^t q_{in} \bar{c}_{in} \, dt - \int_0^t q_{out} \bar{c}_{out} \, dt \]  \hspace{1cm} (7.146)

It is clear that both methods are leading to the same result (lines collapse) from which it can be concluded that the 2DV model is conserving mass.

**Particle size distribution in overflow**

The total concentration of the overflow mixture is plotted versus time in Figure 7.37. This concentration is the sum of the concentrations of all fractions in the calculation. The particle size distribution can be obtained from the concentration distribution because the diameters of the different fractions are known. The median diameter \( D_{50} \) of the PSD's in the overflow can be plotted versus time and compared with the \( D_{50} \)'s determined from the samples taken from the overflow.

![Figure 7.39](image)

*Figure 7.39  Computed and measured \( D_{50} \) of the PSD in the overflow during test 06.*

Figure 7.39 shows that for this test the calculated \( D_{50} \) is smaller than the measured value. The explanation of this difference is the lower computed suspension concentration in the hopper (see Figure 7.34). When the concentration increases, larger particles can remain in suspension due to hindered settling. At the end of the test the median grain diameter \( D_{30} \) decreases. This phenomenon is caused by the decreasing
inflow and outflow concentration during the test. The numerical model predicts the same behaviour.

II. Simulation of Test 05

Test 05 was also used for validation of the numerical model. Two different simulations, with slightly different PSD’s were made. For calculation 1 the same PSD as used for test 06 was used, while for calculation 2 the average value of the samples taken during test 05 were used (see Table 7.5).

![Graph](image)

**Figure 7.40 Measured and computed concentration test 05.**

The computed concentration in the overflow is compared with the measured value in Figure 7.40. The symbols show the concentration determined from the samples. The small difference in PSD between the different computations shows a different relation of the outflow concentration versus time. Calculation 1 was performed with the slightly coarser PSD from test 6 and results in fewer losses during the first phase and larger losses during the final stage. The large overflow concentration at the end of calculation 1 is caused by the continuation of the simulation after the sand bed level has reached the overflow level. The sand bed reached that level during an earlier stage due to the lower losses before that time during calc. 1.

Figure 7.41 shows the cumulative overflow loss (lower panel) and the (specific) discharge versus time (upper panel). Calculation 1 and 2 produced exactly the same cumulative overflow loss at the end of the test, which is caused by the same simulation time of these computations. The total influx of sand was therefore the same for both cases. At the end of the simulation the filling grade of the hopper is also equal, therefore the amount of material in the overflow over that length of time must be equal.
**Figure 7.41** Discharge and cum. overflow loss as function of time (test 05).

Figure 7.42 shows the relationship between the cumulative overflow loss with the hopper load. The hopper load in this case is defined as the total volume of grains (without pore volume) inside the hopper per unit width, which is equal to the sum of the settled sand and the amount of sand still in suspension.

**Figure 7.42** Cum. overflow loss as function of hopper load.

Due to the lower losses during calculation 1 (coarser sand) the hopper is almost completely filled with sand when the cumulative overflow loss is approximately 0.17. After that time (t = 2400 s) the complete influx is “lost overboard” and the cumulative losses increase without a substantial increase in hopper load.
Particle Size distribution in the overflow
The calculated $D_{50}$ of the particle size distribution in the overflow is compared with measurements in Figure 7.43. The computed $D_{50}$ is also lower than the measured value. Here also the difference can be explained from a lower computed suspension concentration.

![Graph showing measured and calculated $D_{50}$ over time](image)

*Figure 7.43  Computed and measured $D_{50}$ of the PSD in the overflow (test 05).*

III. Simulation of test 04
Four experiments were carried out on slightly coarser sand (140 μm). The test with the largest sandflux (test 4) was selected for comparison with the numerical model. The details of this test are given in Table 7.4.

The results are shown in Figure 7.44. It is clear that both measured and computed overflow concentration remained low for the larger part of the simulation. At the final phase the computed outflow concentration increased more gently compared with the experiment. This indicates that diffusion during the computation is larger, which is most likely caused by the higher computed flow velocity in the hopper.

![Graph showing concentration over time](image)

*Figure 7.44  Measured and computed concentration test 04.*
Another difference between the measurements and the computations is the possibility of sand built up above the overflow level. This is possible in reality, but not in the 2DV model as a consequence of the rigid-lid assumption of the free water surface. Build-up of sediment above the overflow level becomes increasingly important when the particle diameter increases. Fortunately, the field of interest for the model is fine sand where losses are high and for that situation the rigid-lid assumption is not too restrictive. In Figure 7.45 the cumulative overflow loss as a function of the load in the hopper is shown. It is clear that the magnitude of the losses is approximated satisfactory, but the relation as a function of time differs due to reasons mentioned above.

Figure 7.45 Cumulative overflow loss as function of hopper load (test 04).

7.13.6 Scaling of the results to prototype values and comparison with 1DV model

Until now, the numerical models (1DV and 2DV) have been used to simulate the model tests. The computations were therefore executed on model scale. The 2DV model was also tested on prototype scale and compared with the 1DV results. The procedure was as follows.

Using the scaling rules of Chapter 4, the results of one of the model tests was scaled to certain prototype dimensions. Test 05 was chosen since this is a typical test. The prototype dimensions were determined using a length scale of five \( (n_L = 5) \). All parameters can be determined by application of the scaling rules (Section 4.3). For instance the length of the hopper becomes: \( L_{\text{proto}} = n_L L_{\text{model}} = 5 \times 12 = 60 \text{ m} \).

The scale for the discharge is \( n_Q = n_L^3 n_u = n_L^{2.5} \), the timescale is \( n_t = \sqrt{n_L} \).

In Table 7.6 the scaled prototype values of test 05 are shown:
<table>
<thead>
<tr>
<th>$Q_{in}$</th>
<th>$\rho_{in}$</th>
<th>$D_{50}$</th>
<th>$L$</th>
<th>$B$</th>
<th>$h_{ov}$</th>
<th>$h_{tan}$</th>
<th>$h_0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>[m$^3$/s]</td>
<td>[kg/m$^3$]</td>
<td>[µm]</td>
<td>[m]</td>
<td>[m]</td>
<td>[m]</td>
<td>[m]</td>
<td>[m]</td>
</tr>
<tr>
<td>5.59</td>
<td>1350</td>
<td>178</td>
<td>60</td>
<td>15.4</td>
<td>11.25</td>
<td>6.25</td>
<td>0.7</td>
</tr>
</tbody>
</table>

Table 7.6  Prototype values based on test 05 for a length scale, $n_L = 5$.

Where $h_0$ is the sediment level in the hopper, $h_{ov}$ the water level at the start of the computation, and $h_{in}$ is the overflow level. The particle size distribution is scaled according to the rule $n_w = n_u = \sqrt{n_L}$ (Not only $D_{50}$, but all fractions). The prototype PSD was determined by taking the following steps:

1. Compute the settling velocity $w_0$ of the eight fractions of the sand used for the computation on model scale (Table 7.5, test 05, Calc. 1).
2. Determine the corresponding prototype settling velocities by multiplication of the model values with the factor $\sqrt{n_L}$.
3. Determine the corresponding grainsizes in prototype.

A prototype value for $D_{50}$ equal to 178 µm was found using this procedure.

Four calculations are compared in Figure 7.46. All results are presented on prototype scale. Two of the calculations were however performed on model scale and two on prototype scale:

1. Computation with the 2DV model on model scale (test 05, Calc 1). The computed overflow concentration is scaled up to prototype scale using the scaling rules. Only the time needs to be scaled, since $n_L = 1$ (model and prototype concentrations are equal).
2. Computation with the 1DV model on model scale. Results are also scaled up.
3. Computation with the 2DV model on prototype scale using the values of Table 7.6.
4. Computation with the 1DV model on prototype scale using the values of Table 7.6.

The measured overflow concentration of test 05 was also scaled-up to prototype scale by transformation of the time using $n_t = \sqrt{n_L}$. The results of the 2DV model were analysed at first. When the 2DV computation on prototype scale is compared with the (scaled-up) 2DV results on model scale, we see that the overflow concentration of both computations is equal until approximately $t = 1500$ s. After that point in time prototype losses are larger until $t = 4800$. Model losses increase rapidly after that point in time because the sand level had reached the overflow level.
Figure 7.46  Computed overflow concentration as function of time (Test 5 scaled up).

It is shown that the scaled-up losses resulting from the model scale computation were smaller than the losses resulting from the computations performed on prototype scale. The difference can be explained by a larger reduction of the sedimentation velocity due to the bed shear stress on prototype scale. The computed losses on prototype scale are smaller than the (scaled-up) computed losses on model scale. On prototype scale the reduction cannot be neglected anymore as can be illustrated by plotting the computed reduction factor, which was recorded during the 2DV-prototype calculation (Calc. 3).

Figure 7.47  Computed reduction factor of the sedimentation velocity as function of the horizontal distance from the inflow position at different times (index = time).

Figure 7.47 shows the computed reduction factor (equation (7.74)) as a function of the horizontal distance in the hopper for different points in time (approximately 1000 s time interval). The largest reduction is present at the start of the loading process when the flow velocity near the bed is largest. Gradually the reduction of the sedimentation velocity decreases with time (reducing flow velocity) and is finally only present in the scour hole below the inflow point. At the end of the loading phase the reduction factor decreases again when the sandbed reaches the overflow level. Flow velocity increases during that final phase due to the decreasing flow depth.
The results of the 1DV calculations are also shown in Figure 7.46. The 1DV model (which does not include the reduction of the sedimentation velocity due to bed shear stress) shows the opposite behaviour compared with the 2DV model. The explanation for this phenomenon is the effect of hindered settling. The settling velocity \( w_0 \) (hence at zero concentration) was scaled from model to prototype. The sedimentation velocity depends on the settling velocity \( w_s \), which includes the influence of sedimentation. The exponent of the hindered settling function decreases with grain size, as was shown in Chapter 3. The reduction of the settling velocity of the coarser prototype sand is therefore less reduced by the concentration, which results in a larger sedimentation velocity (and less overflow losses).

7.14 Concluding remarks testing the 2DV model

The 2DV model provides satisfactory values for the overflow losses. The following differences between the calculated and measured values were observed for the laboratory experiments:
- Computed horizontal flow velocity in the hopper near the bed is larger.
- Computed suspension concentration is lower.
- Computed PSD in overflow is finer.

A possible explanation of these differences is the behaviour of the model in the inflow zone. The vertical velocity in the buoyant jet in the inflow zone is over predicted as was shown in Section 7.12.4 where buoyant jets were simulated. This may be caused by a deficiency of the turbulence model in strong buoyant situations. The higher flow velocity in the jet over predicts entrainment and hence dilution of the inflowing mixture, which results in a lower suspension concentration. Due to the larger entrainment the circulation in the hopper is slightly exaggerated, which enlarges the mixing in the hopper. When calculations are performed with a larger value of the parameter \( c_\mu \) (\( c_\mu = 0.18 \)), like in Section 7.12.4, the results of the model did not improve, because entrainment was enlarged and therefore more mixing in the hopper occurred.

Another possible explanation for the lower velocities measured in the hopper is the presence of air entrained by the plunging mixture. The buoyancy of the jet is decreased by the presence of the air bubbles, which leads to a lower driving force and hence lower velocities. Further investigations are necessary however to prove this possible mechanism.

Computations of the 2DV model on prototype scale demonstrated the importance of the reduction of sedimentation due to the bed shear stress.
Chapter 8
Simulation of prototype measurements taken on board the TSHD “Cornelia”

8.1 Introduction
To test the different models in a prototype situation, measurements were carried out onboard the Trailing Suction Hopper Dredger “Cornelia” owned by Boskalis Westminster Dredging (see Figure 8.1). The dredging area was near IJmuiden (Dutch Coast).

Figure 8.1  TSHD “Cornelia”.
8.2 Hopper Dimensions
The dimensions of the hopper “Cornelia” are:
Length = 52 m
Width = 11.5 m
Volume= 5000 m³

Like most hopper dredgers the shape of the hopper is not rectangular, but has sloping side walls to facilitate dumping of the load. The hopper has the typical silo shape with a keelson. The hopper area is simplified to a rectangular basin with a depth of 5000/(52*11.5) = 8.36 m.

8.3 Operational Parameters
Typical operational parameters during loading (with two suction pipes) were:

Discharge: \( Q = 6 \text{ m}^3/\text{s} \)
Mixture Density: \( \rho_m = 1260 \text{ kg/m}^3 \)
Water density: \( \rho_w = 1021 \text{ kg/m}^3 \)
Porosity (settled sand): \( n_i = 0.46 \)

Samples were taken from the seabed. The following average PSD was determined:

| \( D_{10} \) | \( D_{20} \) | \( D_{30} \) | \( D_{40} \) | \( D_{50} \) | \( D_{60} \) | \( D_{70} \) | \( D_{80} \) | \( D_{90} \) |
| [μm] | [μm] | [μm] | [μm] | [μm] | [μm] | [μm] | [μm] | [μm] |
| 155 | 181 | 200 | 218 | 235 | 254 | 274 | 300 | 330 |

Table 8.1 Average Particle size distribution of the seabed samples.

8.4 Measurements
During a trip the following parameters were measured:
- Discharge into the hopper.
- Density of the mixture.
- Hopper level.
- Overflow level.
- Draught of the ship (from which the loaded tonnage was computed).

In addition to these measurements, which are common on every modern TSHD, the following additional parameters were measured:
- Samples were collected from the overflow mixture to determine the particle size distribution of the overflowing mixture.
- The concentration and horizontal velocity profile in the suspended mixture in the hopper were measured. In Section 8.7.3 these profiles are analysed.
8.5 Overflow loss based on Hopper Load Parameters

The first analysis was based on the dimensionless hopper load parameters $H^*$ and $S^*$ (see Section 4.8.4 and 4.8.5). If these quantities are based on the $D_{50}$ of the PSD, the following values are found respectively: $H^*=0.64$ and $S^*=0.47$.

The following relationship found between $S^*$ and the cumulative overflow loss $OV_{cum}$ can be used as a first rough guess of the cumulative overflow loss (Equation (4.36), Figure 4.9):

\[ OV_{cum} = 0.39 (S^* - 0.43) \]  (8.1)

Application of this equation for $S^* = 0.47$ leads to $OV_{cum} = 1.5\%$. This relationship is based on laboratory experiments, so this value for the overflow loss can be regarded as a lower limit. This was confirmed by the measurements since the measured cumulative overflow loss for the conditions mentioned above was 8%.

8.6 Comparison with 1DV model

The next step is the comparison of the measurement with the results of the 1DV model. All parameters needed for the model are known and given above except for the diffusion coefficient. The order of magnitude for this quantity was estimated using equation (6.21) arriving at a value of $0.05 \text{ m}^2/\text{s}$.

The computed overflow concentration according to the 1DV model is plotted for different values of the diffusion coefficient in Figure 8.2.

![Figure 8.2](image)

*Figure 8.2  Calculated overflow concentration (1DV-model) "Cornelia".*

Figure 8.2 shows that the 1DV model also predicted very low losses (even lower than the measured losses). The different simulations resulted in the following values for the cumulative overflow loss (Table 8.2):
<table>
<thead>
<tr>
<th>Diffusion Coefficient [m²/s]</th>
<th>Cum. Overflow loss [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.005</td>
<td>0.2</td>
</tr>
<tr>
<td>0.01</td>
<td>0.4</td>
</tr>
<tr>
<td>0.05</td>
<td>1.4</td>
</tr>
<tr>
<td>0.1</td>
<td>1.8</td>
</tr>
</tbody>
</table>

**Table 8.2**  
**Cumulative overflow loss for different values of the diffusion coefficient.**

The results of the 1DV model confirmed the first assessment based on the simple empirical relation between $S^*$ and overflow loss. At the expected value for the diffusion the cumulative overflow loss was only of order 1.5 %. This implies that this model underestimates the actual measured losses.

### 8.7 Results of the 2DV Model

In this section the 2DV model was applied to simulate the loading process. In this thesis only one trip was analysed due to the limited variation between the different trips regarding the operational parameters. The results of “trip 7” were used. During this trip two suction pipes were used. At every suction pipe the concentration and velocity were measured during time. These two measurements were combined to get the total discharge and inflow concentration in the hopper. If $Q_{sb}$ and $Q_{ps}$ is the discharge at starboard and portside the total discharge follows simply from the sum of these values. The inflow concentration is the weighted average of the starboard $c_{sb}$ and portside concentration $c_{ps}$ (when $Q_{ps} + Q_{sb} = 0$):

$$c_{in} = \frac{Q_{sb} c_{sb} + Q_{ps} c_{ps}}{Q_{sb} + Q_{ps}}$$  \hspace{1cm} (8.2)

### 8.7.1 Measured and computed overflow losses

In Figure 8.3 the measured inflow concentration and the computed outflow concentration is plotted. The standard settings for the 2DV model were used as indicated in Section 7.13.2. The outflow concentration was not measured onboard.
Figure 8.3  Computed outflow concentration (2D model) Cornelia.

The upper panel of Figure 8.4 shows the measured total discharge in the hopper (which is also an input value for the 2DV model) and the computed overflow discharge (hatched line). The lower panel shows the measured (continuous line) and computed (hatched line) cumulative overflow loss as function of time.

The measured overflow loss was not based on overflow samples, because concentration of these samples is low and therefore the results are unreliable. Instead the measured tonnage of the ship (from the draught), the measured hopper volume and the total measured influx of sand was used to determine the cumulative overflow loss. This method has the disadvantage that at the beginning of the loading process the outcome is very sensitive on the draught and hopper volume and therefore unreliable.

Figure 8.4  Discharge and overflow losses during trip 7.

The results improve however during time as can be seen in the Figure. Before $t = 1500$ s the measured overflow loss varies strongly After that time measured and computed values show the same trend and finally the same cumulative overflow loss is
found. The 2DV model is capable of predicting the measured overflow loss unlike the methods previously presented.

8.7.2 Particle size distribution in overflow

The computation is performed with 10 fractions, hence the (computed) particle size of the sediment in the overflow can be determined and compared with the measured values. Figure 8.5 shows the outcome of this procedure.

![Diagram showing measured and calculated d50 of sediment in the overflow.](image)

*Figure 8.5 Measured and computed d50 of sediment in the overflow.*

Two data points (symbols) indicate the average value of samples during the first stage (0-900 seconds after start overflow) and the final stage of the overflow phase. There is a good agreement between the measured and computed values.

It can be concluded that for this prototype situation the 2DV model provides better results than the 1DV model. This is the result of the reduction of the sedimentation owing to the presence of horizontal transport and the scour crater at the inflow section.

8.7.3 Velocity and concentration profiles

In addition to the measurements discussed above, concentrations and velocities were also measured inside the hopper. These measurements are not common onboard a hopper dredger and therefore special arrangements were needed (Figure 8.6). A guide rail was attached to a vertical pipe in the hopper. Along this rail a nuclear density meter and a velocity probe could be moved in vertical direction by a winch. A computer controlled this winch. By monitoring the hoisting force in the system the location of the bed could be identified. When the probe reached the bed the hoisting force decreased, because the weight of the probe was supported by the bed. As soon as this event took place the winch speed was reversed to avoid burying of the instruments. The lowering and hoisting speed could be regulated. The vertical position was monitored as well, so by combining this information the measured density as a function of time could be transformed to a density profile (density as function of vertical distance). The density is measured between two vertical cones. In one of the cones the nuclear source is placed.
the other contains the receiver. The velocity was measured with an EMS attached beside
the density probe. The EMS can be seen in Figure 8.6 at the end of the light cable. The
measured velocity was logged on a different computer therefore this information could
not be transformed as function of depth as easily.

Figure 8.6  Density and velocity probe.

Concentration profiles are shown for approximately 800 and 1900 s after the start of
loading in Figure 8.7. The computed location of the bed at these moments is also shown
in the same figure. The computed concentration profiles shown on 4 different horizontal
locations can be compared with the measured concentration on the same locations. The
computed profile shapes resemble the shapes observed at the model tests, and are more
uniform over depth compared with the measured profiles. A large difference is present
regarding the location of the sand bed for t = 800 s. This is caused by the presence of
the silo shape of the actual hopper, which is simplified to a rectangular shape in the
model. Owing to this difference the sand level rises much faster during the first stage of
the loading process.
Figure 8.7  Computed and Measured Concentration profiles.

The computed velocity profiles are shown in Figure 8.8. The larger velocities are observed near the bed. In the middle section a return flow is present. Near the water surface a relatively thin, low concentrated layer flowing towards the overflow is present. The computed flow pattern agrees well with the observed flow field of the model hopper sedimentation tests (see Figure 4.17). During the measurements onboard the Cornelia the fast flowing upper layer could be seen as well.

Figure 8.8  Computed horizontal velocity profile.

The measured velocity in the hopper is shown in Figure 8.9. Problems occurred when the signals were converted to physical values in the data-acquisition system onboard, therefore no negative values were logged (although in reality negative velocities were present as was observed on the display of the instrument). The velocity signal is plotted
versus time. The transformation to depth has not been made. The registration shows peaks every time the probe is lowered to the bed. The maximum velocity is approximately $1 - 1.4$ m/s which agrees with the measured value. Doubts however are present regarding the reliability of these signals.

![Figure 8.9 Measured horizontal velocity.](image)

The previous figures have shown an over-all picture of the hopper. In the next figure we focus on the inflow part of the hopper where the scour crater is situated (Figure 8.10).

![Figure 8.10 Computed concentration and velocity in the inflow section.](image)

The left plot shows the concentration and the right plot the value of the velocity. The bed reflects the flow and in the crater a relative high concentration is present. The flow is decelerated and the mixture flows out of the crater as a density current.

### 8.7.4 Segregation in the hopper

It is well known that the particle size varies in a hopper. Closer to the inflow section the sand is coarser than at the other side. This segregation process also occurs during the calculations as shown in Figure 8.11, where a contour plot of the $D_{50}$ of the sand in the
hopper is shown. Closer to the inflow point the sand is coarser, both in the bed as in suspension. No samples were taken from the settled bed during time, so these computational results cannot be compared with measurements.

![Diagram](image)

**Figure 8.11** Computed \( D_{50} \) of the PSD in the hopper.

### 8.8 Concluding Remarks 2D model

Only one prototype test was simulated with the 2DV model. The results show that for prototype situations the 2DV model performs better than the 1DV model. The 1DV model underestimates the losses on prototype scale because the influence of the bed shear stress on the sedimentation velocity is not included in that simple model. This phenomenon was already predicted in Section 7.13.6. The results of the 2DV model are encouraging and justify the effort for developing this complicated model.
Chapter 9
Conclusions and Recommendations

9.1 Introduction
In this thesis the sedimentation process inside a hopper was studied. The existing hopper sedimentation models were evaluated. The influence of concentration, particle size distribution (PSD), clay fraction and air bubbles on the settling velocity of sediment was investigated.
An important part of the research program consisted of experiments:
- Model hopper sedimentation tests were carried out to improve physical insight and obtain high quality data.
- One-dimensional sedimentation tests were performed to investigate the influence of the PSD, turbulence and air bubbles.
- Sedimentation tests were carried out in a closed flume to investigate the influence of the bed shear stress on sedimentation for the circumstance typically encountered in a hopper dredger.
It was recognised that the existing hopper sedimentation models were too limited. Therefore, two new models were developed: a one-dimensional vertical model, based on the advection-diffusion equation and a two-dimensional vertical model, based on the Reynolds Averaged Navier-Stokes equations.
Finally, because it was recognised that high-quality full-scale measurements were not available, hence a measurement campaign was set-up onboard the TSHD “Cornelia”.
The conclusions that have been drawn from the study can be grouped into three categories:

9.2 Physical processes inside the hopper
- Density (buoyancy) effects dominate the flow inside the hopper. In the inflow section the incoming mixture flows vertically towards the bed and forms a scour hole. A density current flows from this location over the bed. From this current sedimentation takes place which leads to a rising sand bed. The velocity distribution is far from uniform due to the presence of the density current. The largest velocities are present near the bed.
- The part of the incoming sediment, which does not settle, moves upward into suspension. At the water surface a low concentrated layer is present which flows
horizontally towards the overflow. Apart from the inflow section the flow in the suspended part of the hopper is basically one-dimensional in vertical direction.

9.3 Hopper sedimentation tests

- The influence of the bed shear stress on sedimentation is underestimated in model hopper sedimentation tests, due to scale effects. One should therefore always be very careful to scale the results of the model tests directly to full-scale values. Nevertheless, these tests executed in the current research programme have significantly improved physical insight, because the attention was focussed on measurements (velocity and concentration) inside the hopper.
- The overflow loss is primarily governed by the dimensionless overflow rate. This parameter is usually expressed as a ratio between two velocities ($H^+$). However, in this thesis is shown that a ratio between sediment fluxes ($S^+$) is more appropriate.
- The influence of the inflow and overflow configuration was negligible for the model hopper sedimentation tests. This conclusion cannot be extended to prototype scale because of scale effects. It is however expected that the influence is of second order compared with the influence of the overflow rate.
- The flow velocity near the bed decreases with bed height (filling) in the hopper, because buoyancy effects govern the flow velocities in the hopper. In existing hopper sedimentation models the flow velocity increases with bed level, because of the assumed uniform or logarithmic velocity profile.
- The sedimentation velocity is not very sensitive on the near bed concentration for fine sands.

9.4 Closed flume sedimentation tests

- The conventional way to include the influence of the bed shear stress on erosion and/or sedimentation is the use of a pickup function. However, when existing pickup function are used, the reduction of the sedimentation due to bed shear stress is overestimated.
- The influence of the bed shear stress on the sedimentation velocity can be modelled with a reduction of the sedimentation in stagnant flow condition. The reduction factor shows a more or less linear relationship with the Shields parameter.
- From the experiments the pickup rate can be determined. It was found that pickup rate decreases with near bed concentration.
- The results of the test confirm the observation that the influence of the bed shear stress on sedimentation was negligible.
9.5 Modelling of the hopper sedimentation process

9.5.1 Existing models

The existing hopper sedimentation models used so far are limited because:

1. All hopper sedimentation models are based on a simplified velocity distribution in the hopper where the inflow and outflow section are simplified or idealised. This implies that the influence of the inflow and overflow constructions (position, level, quantity) on the sedimentation is not accounted for and cannot be simulated.

2. In these models the velocity distribution is assumed to be uniform or logarithmic. In reality density currents develop when high mixture densities are discharged in the hopper. In that case the flow is concentrated near the bed. Velocities in the density current are much higher compared with the uniform or logarithmic distribution. This affects turbulence generation and erosion. The possibility of the presence of density currents in the hopper was already mentioned by De Koning, 1977.

3. The influence of the flow velocity or bed shear stress on sedimentation is modelled in a very simple way.

4. Some models do not include the influence of hindered settling, and when they do the interaction of the different fractions is too simple for high concentrations.

5. Most models do not include the influence of the variation of bed level or the variation of water level during the filling process.

9.5.2 One-dimensional vertical (1DV) hopper sedimentation model

1. It is very important to include the shape of the PSD in the computations. This influence was modelled successfully using Richardson's theory.

2. The 1DV model includes the influence of advection (vertical velocity in the hopper) and diffusion (caused by turbulence). The computed overflow losses are not very sensitive on the value of the diffusion coefficient, which indicates that advection is the dominant process over diffusion.

3. The developed one-dimensional model, based on the observed flowfield, proved capable of simulating the one-dimensional sedimentation tests and the model hopper sedimentation tests quite well.

4. However, this model cannot simulate the process on prototype scale equally well since the horizontal transport in the hopper is not included due to the one-dimensional character of the model. The horizontal transport is accompanied with a horizontal velocity near the bed, which can reduce sedimentation. This phenomenon did not play a major role during the hopper experiments because of the relative small scale.
9.5.3 Two-dimensional vertical (2DV) hopper sedimentation model

1. The developed two-dimensional vertical model based on the Reynolds Averaged Navier-Stokes for mixture flow simulates the sedimentation process quite well. Apart from the possible influence of air bubbles, the model includes the important phenomena of the hopper sedimentation process.

2. The numerical model has been compared with laboratory experiments and a limited number of prototype measurements.

Simulation of the model hopper sedimentation tests:

1. The differences between measured and computed overflow losses are small.

2. The shape of the computed and measured velocity and concentration profiles is similar and show the same behaviour as a function of time, but the quantitative values are different.

3. A possible explanation of these differences is the behaviour of the model in the inflow zone. The computed vertical velocity in the buoyant jet in the inflow zone is too large as was shown in Section 7.12.4, where buoyant jets were simulated. This phenomenon may be caused by a deficiency of the turbulence model in strong buoyant situations. The higher flow velocity in the jet results in too much entrainment and hence dilution of the inflowing mixture, which results in a lower suspension concentration. Due to the larger entrainment the circulation in the hopper is slightly exaggerated which enlarges the mixing in the hopper.

4. Another possible explanation for the lower velocities measured in the hopper is the presence of air entrained by the plunging mixture. The buoyancy of the jet is decreased by the presence of the air bubbles, which leads to a lower driving force and hence lower velocities. Further investigations are however necessary to prove this possible mechanism.

5. Computations of the 2DV model on prototype scale demonstrated the importance of the reduction of sedimentation due to the bed shear stress. The results of the 2DV model on prototype scale are much better than the results of the 1DV model.

Simulation of the prototype measurements:

1. Although only a limited number of prototype tests are available for comparison, it was observed that the 2DV model gives a better prediction of the measured overflow loss than the 1DV model.

2. The better performance can be ascribed to the presence of the horizontal transport and scour hole in the 2DV model. These phenomena are absent in the 1DV model owing to the simplifications in this model.
9.6 Recommendations

1. The erosion-sedimentation boundary condition used in the 2DV model is a purely empirical relationship based on the closed circuit experiments. The application of this condition in the model leads thus far to good results, but a proper physical foundation of the relationship is needed to verify that it can be applied for all scales.

2. Extra experiments are needed to calibrate the turbulence model in the inflow section where the buoyancy of the jet dominates. It is suspected that the entrainment is too large in this section. The influence of air can be studied as well during these tests.

3. The influence of air can be included in the model by introducing air bubbles as buoyant fractions. The amount of air and bubble size distribution can be based on empirical relationships developed for plunging jets.

4. In this thesis the bed boundary condition is implemented as a reduction factor on the settling flux, which is an acceptable approach for hopper sedimentation application where sedimentation is the dominant mechanism since the flow is always overloaded with sediment. For more general application the boundary condition should be written as a pickup function. In that case the erosion of the bed can also be modelled.

5. The pickup function should include the influence of the near bed concentration.

6. For the boundary condition at the water surface a rigid lid approach was used. Applying a real free water level condition (with zero pressure) is recommended. This leads to a better performance of the pressure correction method and makes it possible to build up the sand bed above the overflow level. The latter can occur for coarser sand or low overflow rate.

7. The present 2DV model can be extended to a 2DV width-averaged model to include the influence of the hopper geometry (silo shape) in the model.

8. More prototype measurements must be undertaken for further validation of the 2DV model for different PSD and hopper geometry and size.
References


Anderson, J.D., 1995, ”Computational fluid dynamics; the basic with applications”, New York McGraw-Hill.


220


222


Rhee, C. van, Talmon, A.M., 2000, "Entrainment of sediment (or reduction of sedimentation) at high concentration", 10th Int. Symposium on Transport and Sedimentation of Solid Particles, Wroclaw, Poland.


Spengen, J. van, 2002, “TU Delft, Faculteit Civiele Techniek en Geowetenschappen,


224
Stelling, 2000 "personal communication"


Zijlema, M., 1994, Finite volume computation of 2D incompressible turbulent flows in general co-ordinates on staggered grids”, Technical University Delft,

Appendix A - List of Symbols

Roman symbols

$A$  Hopper area
$A_g$  Projected surface of a grain
$A_s$  Spreading coefficient
$a$  Coefficient
$B$  Width
$B_s$  Spreading coefficient
$b_y$  Coefficient analytical solution advection-diffusion equation
$C$  Discharge coefficient
$C_D$  Drag coefficient
$C_m$  Added mass coefficient
$c$  Volume concentration
$c_{re}$  Reference concentration
$c_b$  Near bed concentration
$c_i$  Concentration of fraction $i$
$c_{in}$  Inflow concentration
$c_t$  Total concentration (sum of all fractions)
$c_o$  Overflow concentration
$c_{max}$  Maximum bed concentration
$c_{m}$  Coefficient for the $k-\varepsilon$ model
$c_{re}$  Coefficient for the $k-\varepsilon$ model
$c_{2e}$  Coefficient for the $k-\varepsilon$ model
$c_{re}'$  Coefficient for the low-Reynolds $k-\varepsilon$ model
$c_{re}''$  Coefficient for the low-Reynolds $k-\varepsilon$ model
$c_{2e}'$  Coefficient for the low-Reynolds $k-\varepsilon$ model
$D$  Grain diameter
$D$  Width of a nozzle
$D_{cr}$  Threshold grain diameter
$D_h$  Hydraulic diameter
$D_s$  Dimensionless grain diameter
$d_{in}$  Height of inflow zone
$d_b$  Bubble diameter
$E$  Pickup mass flux
$E_r$  Expansion ratio backward facing step
$F$  Froude number
$F_i$  Internal Froude number

$f$  Darcy-Weisbach friction coefficient

$f_{\mu}$  Damping function low-Reynolds $k-\varepsilon$ model

$f_i$  Damping function low-Reynolds $k-\varepsilon$ model

$f_{m}$  Damping function low-Reynolds $k-\varepsilon$ model

$G_{h}$  Total volume of sediment loaded in hopper

$G_{p}$  Total volume of sediment pumped into hopper

$g$  Gravitational acceleration

$g_{hl}$  Sediment flux loaded in hopper

$g_{o}$  Sediment flux in overflow level

$g_{p}$  Sediment flux pumped into the hopper

$H$  Water depth

$H_{sl}$  Height of settles sediment

$H^*$  Dimensionless overflow rate parameter

$h$  Water level above overflow

$h_{n}$  Water level in the hopper

$h_{0}$  Thickness of density current in 1DV model

$h_{ov}$  Overflow level

$I_{T}$  Turbulence intensity

$i$  Hydraulic gradient

$i_{eq}$  Equilibrium slope

$j_{n}$  Coefficient

$K$  Viscosity parameter

$K_{1}$  Jet spreading coefficient

$K_{2}$  Jet spreading coefficient

$k$  Turbulent energy

$k$  Permeability

$k_{e}$  Roughness height according to Nicuradse

$L$  Length

$\ell$  Length scale of turbulent motion

$\ell_{m}$  Mixing length

$M_{so}$  Mass of solids in the hopper

$N$  Bagnold number

$n$  Scale factor

$n_{0}$  Porosity

$O$  Wetted perimeter

$OV$  Overflow flux

$OV_{cum}$  Cumulative overflow loss

$P$  Production term in $k-\varepsilon$ model

$P_{b}$  Buoyancy Production term in $k-\varepsilon$ model

228
Pe  Péclet number
p  Pressure
p_0  Hydrostatic pressure
p_d  Hydrodynamic pressure
p'_0  Cumulative percentage corresponding with \( v_0 \)
p_r  Cumulative percentage corresponding with \( w_{s,cr} \)
Q  Discharge
q  Specific discharge
q_a  Air entrainment rate per unit width
q_s  Inflowing sediment flux
R  Reduction factor for sedimentation velocity, Hydraulic radius
R_y  Reynolds number
Re_r  Reynolds number
Re_p  Particle Reynolds number
r'_g  Removal ratio of a fraction
r_r  Total removal ratio
r_i  Effect of turbulence on removal ratio of a fraction
Ri  Richardson number
S  Settling flux
S_n  Net settling flux
S_0  Settling flux at zero flow velocity
S_y  Sedimentation coefficient
S^*  Dimensionless overflow rate parameter
s  Sediment (mass) flux
s_s  Sediment flux in overflow
s_{sed}  Settling mass flux
St  Stokes number
T  Transport parameter
t  Time
t_L  Loading time
U  Depth averaged horizontal flow velocity
U_0  Inflow velocity
U_H  Depth averaged horizontal flow velocity
u  Horizontal velocity
u_1  Jet impingment velocity
u_o  Velocity in flume before valve is closed, exit velocity from nozzle
u_s  Friction velocity
u_f  Front speed of density current
u_a  Flow velocity in overflow
u_m  Centreline velocity in a (buoyant) jet
$u_s$ Scour velocity  
$V_r$ Volume of a grain  
$V_b$ Volume of settled sediment in control volume near the bed  
$V_s$ Volume of suspension in control volume near the bed  
$v_{in}$ Inflow velocity  
$v_0$ Overflow rate  
$v_s$ Slip velocity  
$v_{sed}$ Sedimentation velocity  
$v_{sed,0}$ Sedimentation velocity at zero flow velocity  
$v_w$ Vertical fluid velocity, Vertical velocity of a grain  
$v_z$ Vertical velocity of a grain  
$w$ Vertical (mixture) velocity  
$w_0$ Settling velocity single grain  
$w_s$ Settling velocity  
$w_{s,cr}$ Settling velocity corresponding with scour velocity  
$w_H$ Vertical velocity of water surface  
$x$ Horizontal co-ordinate  
$Y$ Distance to the wall  
$z$ Vertical co-ordinate  
$z_u$ Reference height  
$Z$ Suspension parameter  

Greek symbols

$\alpha_n$ Coefficient  
$\beta$ Coefficient  
$\Delta$ Specific sediment density $(\rho_s-\rho_w)/\rho_w$  
$\Delta_k$ Specific sediment density for grain $k$  
$\varepsilon$ Dissipation of turbulent energy  
$\varepsilon$ Air concentration  
$\varepsilon_0$ Brownian diffusivity  
$\varepsilon_z$ Vertical diffusion coefficient  
$\delta$ Factor  
$\phi$ Factor  
$\phi$ Scalar quantity  
$\phi_p$ Dimensionless Pickup rate  
$\Gamma$ Diffusion coefficient  
$\lambda$ Linear concentration, radius ratio, Kolmogoroff length scale  
$\gamma$ Coefficient  

230
\( \dot{\gamma} \) Shear rate
\( \eta \) Loading efficiency, Dynamic viscosity, ratio between production and dissipation
\( \eta_0 \) Coefficient, Dynamic viscosity of clear water
\( \eta_m \) Average loading efficiency
\( \kappa \) Von Kármán constant
\( \mu \) Coefficient in scour velocity expression
\( \nu \) Kinematic viscosity
\( \nu_e \) Eddy viscosity
\( \theta \) Shields parameter
\( \theta_c \) Critical Shields parameter
\( \theta_0 \) Shields parameter at critical velocity (deposition limit)
\( \rho \) Mixture density
\( \rho_0 \) Density of ambient water
\( \rho_d \) Density difference with water
\( \rho_{mwp} \) Mixture density
\( \rho_s \) Density of sediment
\( \rho_w \) Density of water
\( \sigma \) Schmidt number
\( \sigma_t \) Transformed vertical co-ordinate
\( \sigma_k \) Schmidt number for the \( k - \varepsilon \) model
\( \sigma_s \) Schmidt number for the \( k - \varepsilon \) model
\( \tau \) Shear stress
\( \tau_B \) Bingham yield stress
\( \tau_b \) Bottom shear stress
\( \tau_{wp} \) Average wall shear stress computed from pressure gradient
\( \tau_{cor} \) Approximated average wall shear stress
\( \tau_p \) Particle response time
\( \tau_n \) Hydrodynamic time scale
\( \tau_y \) Yield stress
\( \omega \) Vorticity
\( \xi \) Dimensionless depth
\( \xi_n \) Dimensionless water level above overflow level
\( \psi \) Shape factor
\( \psi \) Stream function
Appendix B - One Dimensional Sedimentation Tests

The one-dimensional sedimentation tests (Klerk et. al., 1998, Runge & Ruig, 1998) were carried out in a tube with an inner diameter of 0.3 m and height of 1.5 m. Inside the tube a grid was placed, which was fixed to the walls of the column. By rotation of the column (with grid) turbulence inside the column could be generated and the sedimentation of sand in turbulent conditions could be studied as well. It is more common to generate turbulence by oscillating the grid into vertical direction inside a stationary column. In this case this approach could not be used due to the projected high initial sand concentration and the resulting high level of settled sediment inside the column after a test. The grid would in that case be forced to move through the sand-bed that would lead to large forces on the grid and most probably destruction of the grid.

![Sedimentation column with grid](image)

Figure B.1  Sedimentation column with grid

The total experimental set-up consisted of a mixing vessel, where a sand-water mixture was prepared, the sedimentation column, a pump and some tubing to pump the mixture form the mixing vessel towards the sedimentation column. The column was filled through a pipe connected to the bottom within a short time.

In the column sand concentration was measured with 2 point conductivity probes at twelve different vertical positions, so a good impression of the concentration vertical could be formed. On four different vertical positions taps were placed to withdraw a mixture sample from the column during a test. These samples were used to determine
the concentration and the particle size distribution. Tests were carried out on uniform sands with grain diameter $D_{50}$ of 80, 160 and 270 $\mu$m and graded sand with a $d_{50}$ of 160 $\mu$m ($d_{10}$ is 85 $\mu$m, $d_{90}$ is 500 $\mu$m).

Volume concentration was varied up to 45 %. (by volume)
Appendix C – Grid Transformation

The grid transformation method is explained below using a simple example (see Figure C.1). In this figure a cartesian grid is shown. The water surface coincides with the top of the grid (z=0). A curved line represents the location of the sand bed ( h is the water depth). The grid lines do not coincide with the sand bed shown in the figure. A new grid can be constructed where the bed coincides with the grid lines when a grid transformation is performed.

The independent variables in physical space (x,z,t) is transformed to a new set of variables in transformed space (ξ,σ,τ). When the following so-called sigma transformation is defined (Phillips, 1957):

\[ ξ = x \]
\[ σ = \frac{z}{h(x)} \]
\[ τ = t \]  \hspace{1cm} (C.1)

![Diagram of Cartesian Grid with curved bed geometry](image)

Figure C.1  Cartesian Grid with curved bed geometry.

The transformed domain is cartesian with σ = 0 at the water surface and σ = -1 at the bed (see Figure C.2). Like the domain the Partial differential equations have also to be transformed to this space.
Figure C.2  Transformation of co-ordinates.

The consequences is illustrated below using the continuity equation:

\[ \frac{\partial u}{\partial x} + \frac{\partial w}{\partial z} = 0 \]  \hspace{1cm} (C.2)

The independent variables \( u \) and \( w \) appear in this equation as derivatives, so a transformation for the derivatives is needed. In general this can be written:

\[ \frac{\partial}{\partial x} = \frac{\partial}{\partial \xi} \frac{\partial \xi}{\partial x} + \frac{\partial}{\partial \sigma} \frac{\partial \sigma}{\partial x} \]
\[ \frac{\partial}{\partial z} = \frac{\partial}{\partial \xi} \frac{\partial \xi}{\partial z} + \frac{\partial}{\partial \sigma} \frac{\partial \sigma}{\partial z} \]  \hspace{1cm} (C.3)

The derivatives of the transformed co-ordinates with respect to the physical co-ordinates can be derived:

\[ \frac{\partial \xi}{\partial x} = 1 \]
\[ \frac{\partial \sigma}{\partial x} = -\frac{\sigma dh}{h \, dx} \]
\[ \frac{\partial \xi}{\partial z} = 0 \]
\[ \frac{\partial \sigma}{\partial z} = \frac{1}{h} \]  \hspace{1cm} (C.4)

Using these expressions we can transform the continuity equation. (using \( \xi = x \)):

\[ \frac{\partial u}{\partial x} - \frac{\sigma dh}{h \, dx} \frac{\partial u}{\partial \sigma} + \frac{1}{h} \frac{\partial w}{\partial \sigma} = 0 \]  \hspace{1cm} (C.5)
It is customary to use a transformed expression for the vertical velocity as well. For a stationary situation this leads to:

$$\omega = w - u \sigma \frac{dh}{dx}$$  \hspace{1cm} (C.6)

By substitution of this expression the continuity equation can be simplified to:

$$\frac{\partial (hu)}{\partial x} + \frac{\partial \omega}{\partial \sigma} = 0$$  \hspace{1cm} (C.7)

The only unknowns in this equation are u and \(\omega\) since h only depends on the geometry of the physical domain. The equation can thus in principle be solved. All other PDE’s (momentum, sediment transport and \(k - \varepsilon\)) must be transformed likewise.
Appendix D – Influence of specific density correction for uniform sediment

In Chapter 3 the specific density correction according to Selim (1983) was described. This method can produce better results when the settling of multi-sized mixtures is computed for mixtures where a large difference in sizes (or density) is present (Selim, 1983). The settling velocity is however possible underpredicted when the particle size distribution is more uniform. Suppose for instance that the settling velocity of almost uniform sediment is analysed. Particle size ranges from 90 – 110 μm. The one-dimensional settling velocity of a 20% sediment-water mixture is calculated for five different situations:

1. A monosized mixture with a particle diameter of 100 μm.
2. A mixture of 2 sizes with diameters of 95 μm (50 %) and 105 μm (50 %) respectively. The density correction of Selim is not used during the calculation.
3. A mixture of 2 sizes with diameters of 95 μm (50%) and 105 μm (50%) respectively with density correction.
4. A mixture of 4 sizes: 92.5 μm, 97.5 μm, 102.5 μm and 107.5 μm respectively. The total amount of sediment is equally distributed over the different fractions. The density correction of Selim is not used during the calculation.
5. A mixture of 4 sizes: 92.5 μm, 97.5 μm, 102.5 μm and 107.5 μm respectively. The total amount of sediment is equally distributed over the different fractions. The density correction of Selim is used during the calculation.

A column is filled with a mixture level is 2.0 m. Settling takes place and after some time a sandbed had formed in the lower section of the column. The influence of the different particle size distributions and the density correction can be shown when the height of the settled is plotted as function of time in Figure D.1. In this Figure five lines are shown. The height of the bed for the multisized mixtures as function of time is independent of the number of fraction for the calculations where the specific density correction was not used. The calculation with specific correction applied shows a lower increase in bed height as function of time. The outcome is also depends on the number of fraction used in the computation. When this method is used the results therefore depend on how the particles size distribution is approximated. The differences increase with concentration since the density of the suspension also increases with concentration. The method should therefore only be used for well-graded sands.
Figure D.1  Height of settled bed as function of time.
Acknowledgements

This study has been conducted at the Delft University of Technology, Faculty of Design, Engineering and Production, section Dredging Technology in co-operation with the section of Coastal Engineering of the faculty of Civil Engineering and Geosciences. It was supported by the Vereniging van waterbouwers in Bagger-, Kust- en Oeverwerken (VBKO) and the Dutch ministry of economic affairs (SENDER).

I would like to thank prof. ir. W.J. Vlasblom, prof. ir. C. d’Angremond, dr. ir. J.C. Winterwerp, dr. ir. C. Kranenburg and prof. dr. ir. J.A. Battjes for there valuable comments and support. I would like to thank dr. ir. Arno Talmon. His guidance and stimulation were indispensable for the completion of the research program and this thesis.

I wish to thank all colleagues at the Delft University of Technology and especially ir. Joost Joustra and ir. Marco den Burger for many stimulating discussions. I would like to extend my gratitude to my colleagues at Ballast Ham Dredging who were always interested in the progress of the PhD-project. The support of ir. Hans Oostinga and the other colleagues of the RDE department is gratefully acknowledged. My presence on the University for two days per week for a long time resulted often for them in a heavier workload. I would like to thank Jim Anderson who had to read the complete thesis in order to take care of the linguistic aspects.

The experiments carried out for the research program would not have been possible without the help of many (former) students. Within the framework of their MSc thesis they contributed to the research programme. I like to mention: Dingena Schott, Ad van Es, Bastiaan Ruig, August Runge, A. de Klerk, J.C. Meulepas, Petar Lubking, Jarda van Spengen, Georges Teheux, Andries Gerritsen, Steven Zuiderwijk and Edo Hoekstra.

I would like to thank prof. dr. ir. G. Stelling for his encouragement and advise on the numerical implementation of the $k-e$ model.

Very important to me was the backing of ir. Bart van der Schrieck. He brought the possibility of the part time promotion to my attention and convinced the board of directors of my company to give me opportunity.

I would like to thank the VBKO for their financial support, and especially Hendrik Bijnsdorp. His patience when my progress reports were again overdue is very much appreciated. I would like to thank Royal Boskalis Dredging for making available the prototype measurements of the “Cornelia”. Dr. ir. Florian van Kats performed a great job in the preparation of these measurements and post processing of the results.

I like to thank my parents for providing me with the education that enables me to do the work I like. And Finally, my dear Monique, Lotte and Ewoud, I want to thank you for
your enthusiasm, support and understanding during a five year period when I spent the largest part of my time on this PhD.
Curriculum vitae
