On Drops and Turbulence in Nuclear Fuel Assemblies of Boiling Water Reactors

DIANA CARAGHIAUR GARRIDO

Doctoral Thesis
Stockholm, Sweden 2012
Akademisk avhandling som med tillstånd av Kungl Tekniska högskolan framlägges till offentlig granskning för avläggande av teknologie doktorsexamen i energiteknik fredagen den 14 december 2012 klockan 10:30 i Svedbergssalen (FD5), AlbaNova Universitetscentrum, Roslagstullsbacken 21, Stockholm.

© Diana Caraghiaur Garrido, December 2012

Tryck: Universitetsservice US AB
Abstract

The study aims to develop the understanding of the mechanistic-type approach to quantify drop deposition in nuclear fuel assemblies of Boiling Water Reactors. This includes the effect of spacers. Spacers have a complex geometry to serve their purposes, but optimization of them alone can improve the thermal limit parameters in nuclear fuel assemblies. Thus, a mechanistic model might prove useful to increase the safety of the reactor as well as economic competitiveness of the nuclear power plant.

In this thesis, measurement techniques, such as mobile pressure rod and Laser Doppler Velocimetry are developed and tested to provide local data of the flow around spacers. It is shown experimentally that the effect of spacer on the flow differs depending on the placement of the subchannel in the rod bundle. Partly, because the spacer part differs, but also due to a global velocity profile development. Very few studies in the literature indicate this effect. It is shown that single subchannel models using Computational Fluid Dynamics (CFD) can predict the average velocity increase downstream of the spacer; however, they are not capable of calculating the spacer effect on turbulence parameters. The single subchannel CFD model has limited capability to predict the pressure development inside the spacer part, mainly because cross-flows are not taken into consideration.

The deposition of drops in annular two-phase flow is still a scientific challenge. Only empirical correlations are used nowadays to quantify this process. Empirical coefficients are needed for each spacer type to calculate the deposition increase due to obstacle. The discussion about the deposition starts with the phenomenological description. The important input parameter, namely drop size, is carefully analysed, and a new correlation is proposed to calculate the mean drop diameter. The correlation is constructed on a larger experimental data base. Lagrangian Particle Tracking model is tested in its capability to calculate deposition. Additionally, a Eulerian-type model is developed and tested. Turbulent parameters of drops are tightly related to the turbulence of the gas phase and the inertia of the drops. Several approaches are discussed about how to calculate the root-mean-square fluctuating velocities of drops. Both, Lagrangian Particle Tracking and the Eulerian-type of models show good capability in calculating the obstacle effect on deposition, providing improvements are made in prediction of drop size. The effect of increased drop concentration plays a large role and it must be taken into consideration if good quantitative approaches are envisaged.
Acknowledgements

I would like to express my gratitude to the people who helped and supported me during the preparation and writing of the thesis. I wish to thank my supervisor Prof. Henryk Anglart for giving me the opportunity to work with the subject of the thesis within international projects and for offering me large freedom. I also would like to thank Dr. Wiktor Frid, who initiated and supervised the experimental part. Stellan Hedberg was an excellent help in the laboratory. Dr. Nils Tillmark is acknowledged for his help in calibrating the measurement techniques. Most of all I wish to thank Dr. Carl Adamsson in sharing his knowledge on drop dynamics, mathematics and any other challenging subject. The proof-reading of the thesis was performed by Dr. Jan Dufek.

Special thanks goes to Rafael Garrido for the original design of the cover picture. Many thanks to my past and present colleagues and friends from the Nuclear Reactor Technology Division: Ionut, Carl, David, Carsten, Roman, Jan and Maria for help in different matters, for interesting discussions on history, politics, religion, gender equality and other exciting subjects and for creating a stimulating working environment.

The financial support from the European FP6 NURESIM and FP7 NURISP projects, as well as from the Swedish Radiation Safety Authority (SSM) is gratefully acknowledged.

In the end I have to thank the people who rather felt the progress of my work. I am grateful to my son, Simon, for keeping asking me how many pages I still have to write in ‘that book’. Great motivation! I cannot thank enough my husband, Benito, for the enormous support. I would not be able to finish it without you. I feel largely indebted to my family in Moldova who has always been supportive.

Thank you!
To focus, easy to say; you need to know on what. Or, you don’t know it unless under the impulse of passion. But passions cannot be studied. The problems can, but problems do not mean anything.

EMIL CIORAN
List of Publications

Included Papers


IV. D.Caraghiaur and H.Anglart. Drop deposition in annular two-phase flow with Lagrangian Particle Tracking. *submitted to Nuclear Engineering and Design*

V. D.Caraghiaur, C.Adamsson and H.Anglart. A model for inertial drop deposition suitable to predict obstacle effect. *submitted to Nuclear Engineering and Design*

Papers not included

i. D.Caraghiaur, W.Frid and N.Tillmark. Detailed pressure drop measurements in single- and two-phase adiabatic air-water turbulent flows in realistic BWR fuel assembly geometry with spacer grids. In *The 6th International Conference on Nuclear Thermal Hydraulics, Operations and Safety (NUTHOS-6) Nara, Japan, October 4-8, 2004*

software applied to CHF investigations. *Science and Technology of Nuclear Installations*, Article ID 214512, 2009

iii. D.Caraghiaur and H.Anglart. Verification of Discontinuous Random Walk Lagrangian Particle Tracking as a tool to model deposition in annular flow. In *the 17th International Conference on Nuclear Engineering (ICONE17)*, Brussels, Belgium, July 12(€”16, 2009


Contents

1 Background 1

2 Flow around spacers 5
   2.1 Problem statement ................................. 5
   2.2 Experimental arrangement .......................... 6
      2.2.1 Test loop ................................... 6
      2.2.2 Measurement technique ...................... 8
      2.2.3 Accuracy ................................. 9
   2.3 Experimental findings ........................... 10
   2.4 Computational Fluid Dynamics predictions 14

3 Drop deposition. Influence of obstacle on deposition 19
   3.1 Problem statement ................................. 19
   3.2 Phenomenological description .................... 20
      3.2.1 Process of deposition ...................... 20
      3.2.2 Drop size ................................. 24
   3.3 Lagrangian Particle Tracking .................... 28
   3.4 Eulerian approach ............................... 32
   3.5 Influence of obstacle on deposition ......... 38

4 Concluding remarks 41

Bibliography 43
Chapter 1

Background

In a nuclear reactor the heat from the nuclear fission has to be extracted and transformed into mechanical energy of a turbine. The extracting medium is usually a liquid, a gas, or mixture of liquid and gas, in the form of boiling water, for example. Water has been historically the most used heat extracting medium in a nuclear reactor. Mostly because it is a common substance found in nature, it is cheap and it is well studied. At least 98% of all the functioning reactors in the world are cooled with water, as shown in Fig. 1.1.

An efficient way to extract the heat is by boiling the water. Indeed, the share of the Boiling Water Reactors (BWR) in the world is large, amounting to 21%, seen in Fig. 1.1. In BWR conditions of high mass and heat fluxes the boiling of water takes on different forms. Starting from the heating of the liquid water up to its saturation temperature enough premises are created for steam bubbles to appear. The bubbles grow bigger and start to coalesce forming steam slugs. In this regime the liquid water still wets the heated surfaces and envelopes the slugs with bridges in-between. As the mixture is further heated, the liquid bridges break up. What remains is liquid water on the surface of the walls and steam in the core of the channel. The high-flow-rate steam creates waves and ripples on the surface of the liquid film, from which small drops are created. These drops move in the steam until they deposit back to the liquid film, or eventually evaporate. A major depletion of the liquid film by drop formation or evaporation develops into a boiling crisis, called dryout. This occurs when the heated surface is in contact for a longer time with steam only. The result is a drastic increase in surface temperature. The sudden increase in temperature is due to the fact that steam is a much worse heat transfer medium than water. In a nuclear reactor the dryout can result in fuel cladding damage with subsequent radioactive release, which must be avoided. Thus, dryout is one of the major design and operational parameters for Boiling Water Reactors.

The BWR fuel rod bundle has a rather complex geometry and power distribution. To design and operate BWR fuel, techniques capable to predict the critical power performance must be at hand. Three approaches have been used in practice.
The first approach is the limit line, which represents a conservative lower limit to the appropriate boiling crisis data, such that no data points fall below this line (Lahey and Moody, 1993). The limit lines have been constructed in the flux-quality plane on the basis of single and multirod experiments. They are valid in the experimental range they were constructed on. The limit line approach allows for safe BWR operation, but does not capture the axial power distribution effect and, thus, the axial location of the boiling crisis is usually predicted incorrectly.

The second approach to predict the critical power performance is the critical quality-boiling length correlation. It usually has the following form (Lahey and Moody, 1993):

$$\langle x_c \rangle_c = f(L_B, G, p, L_H, D_q, R)$$  (1.1)

where $p$ is the system pressure, $G$ is the mass flux, $D_q$ is the thermal equivalent diameter, $R$ is the generalized local peaking factor, $L_B$ - is the boiling length and $L_H$ is the heated length.

The correlation, similar to the limit line, uses cross-sectional bundle-averaged parameters rather than local parameters. The generalized local peaking factor $R$ is synthesised to quantify the thermal performance of the critical rod on the neighbouring rods. These correlations are highly reliable in many situations, but the
lack of physical modelling makes them unreliable for complicated three-dimensional power distributions.

The desire to predict accurately the critical thermal performance and the distribution of phases in fuel rod assemblies resulted in the formation of subchannel analysis. The subchannel analysis is the third approach. Within this technique the rod bundle is divided into a number of imaginary subchannels. Once the subchannel grid is established, the appropriate two-phase conservation equations can be integrated to yield the axial parameters of interest in the rod bundles.

The subchannel analysis codes have a much bigger degree of freedom than the limit line approach or the dryout correlation, and have the potential to predict the axial thermal performance of a fuel bundle. However, the models incorporated into the subchannel codes still reflect high levels of empiricism. In BWR most empirical correlations describe the effect of spacers on the flow and heat transfer, and mass exchange related to drop dynamics.

The mechanical function of spacers is to limit the axial and radial movement of fuel rods, as well as their vibration. The spacers, besides contributing to the pressure drop across the core, have a significant effect on the heat transfer. Overall fuel performance can be improved by changes in spacer design, without changes in the remainder of the fuel assembly. To minimize the flow blockage and provide enough support to the fuel bundle structure, spacer grids have been transformed over the years to include complex geometrical details. Thus, the subchannel-averaged model is not able to capture the effect of the geometry complexity. Instead, empirical coefficients are obtained to quantify the spacer effect on the flow and heat transfer. In this way, the validity of the models is a function of a particular spacer design.

The dryout occurs when the liquid film disappears at the heated wall. Thus, the phenomenon is best described by the mass balance equation for the liquid film. What adds to the mass of the liquid film is the deposited drops. Empirical correlations are used in the subchannel analysis to calculate the deposition rate. These correlations predict the deposition rate with acceptable accuracy in the parameter range of the correlation database. It was shown experimentally, i.e. Okawa et al. (2011) and Damsöhn (2011), that deposition increases significantly downstream of a flow obstacle. This is one reason why spacers in a nuclear fuel assembly of BWR have a significant effect on the heat transfer, and in particular on the dryout occurrence. Damsöhn (2011) observed that the behaviour of obstacles in regard to drop deposition is difficult to predict with simple parameters, as the dependence on the flux ratio, gas density and obstacle shape are strongly nonlinear. This effect is included in the models by the use of empirical coefficients (Adamsson and Anglart, 2010).

The Computational Fluid Dynamics (CFD) is a much studied and promising technique. It has been shown to have good capability to predict the single-phase flow parameters in simple geometries. However, it is not yet a thoroughly validated technique for complex geometries, such as nuclear fuel assemblies. With the increase in computer capacity this becomes less of an issue, because less modelling - more first principles can be applied. In the case of two- or multi-phase flow calculations,
the CFD method is not mature and is at the stage of closure laws development. Nevertheless, it has the potential to become the design tool for the nuclear fuel assemblies.

Thus, the primary focus in this thesis is to analyze tools, techniques and models, which are able to quantify the spacer effect and drop deposition towards a more mechanistic description. Experimental techniques, such as Laser Doppler Velocimetry and a mobile pressure rod, are tested in their capability to obtain detailed data around the spacer of a nuclear fuel mock-up. Chapter 2 presents the summary of this analysis. It also discusses the ability of Computational Fluid Dynamic codes to calculate the flow around the spacer. Chapter 3 is devoted to drop deposition, including the phenomenological analysis of the deposition process and the ways it can be modelled. Chapter 4 presents the concluding remarks.
Chapter 2

Flow around spacers

2.1 Problem statement

A mechanistic model able to predict the flow behaviour around complex geometrical parts, such as spacers in nuclear fuel bundle, requires detailed measured flow data for its validation. Rowe and Chapman (1973) performed experiments where they showed that turbulence intensity for an internal subchannel significantly varied with distance downstream from the spacer. The high intensity observed at the trailing edge of the spacer rapidly decayed to values below the fully developed condition at a moderate distance from the spacer. Further downstream, the level of intensity increased to nearly twice the level of intensity of fully developed flow, before finally decaying to the developed state. In contrast to the internal subchannel, the wall subchannel had shown high turbulence intensity near the trailing edge of the spacer and a monotonic decay downstream towards a fully developed condition. Yang and Chung (1998) and Nagayoshi and Nishida (1998) have also conducted experiments of turbulence parameters downstream of a spacer. They have observed the behaviour of sudden increase of turbulence intensity at the trailing edge of the spacer followed by a monotonic decay - a characteristic wall subchannel behaviour in the study by Rowe and Chapman (1973). To the author’s knowledge no other studies explicitly indicate the internal subchannel behaviour observed by Rowe and Chapman (1973). Rowe and Chapman (1973) stated that a firm explanation of a "quiet region" downstream of the spacer is not available and it could possibly be drawn from the analysis of turbulence anisotropy. They presupposed that the region of low intensity is due to the transfer of turbulence energy from the axial component to the lateral one. However, another plausible explanation is that the turbulence transfer is coupled to the velocity profile formation in the domain of the rod bundle. Recent experimental studies conducted by Dominguez-Ontiveros et al. (2010) confirm that besides velocity profiles in each subchannel, there is an overall rod bundle profile, illustrated in Fig.2.1. To confirm this behaviour more studies are needed.
The effect of the spacer on turbulence parameters is usually quantified by an exponential decay, determined by the blockage ratio of the spacer $\theta$. For example, Nagayoshi and Nishida (1998) proposes the following equation:

$$\frac{u_{sp}'}{u_0'} = 1.0 + 6.5\theta^2 e^{-0.27(z/D_h)} \quad (2.1)$$

where $u_{sp}'$ is the root-mean-square of the fluctuating velocity downstream of the spacer, $u_0'$ is the root-mean-square of the fluctuating velocity upstream of the spacer, $z$ is the axial distance measured from the trailing edge of the spacer and $D_h$ is the hydraulic diameter of the channel/subchannel.

This kind of description, as indicated in Eq.(2.1) possibly quantifies well the spacer effect of the wall-type subchannel; however it cannot capture the behaviour of the flow downstream of the spacer in internal subchannels.

### 2.2 Experimental arrangement

#### 2.2.1 Test loop

The experimental rig, schematically shown in Fig.2.2, consists of a test section, a water reservoir, a 15 kW centrifugal pump, flow control valves and stainless steel piping, forming a closed loop. The water flow is measured by an electromagnetic
2.2. EXPERIMENTAL ARRANGEMENT

flow meter downstream of the pump. From the pump the water enters the plenum connected to the lower part of the vertical test section. The plenum contains a honeycomb flow rectifier suppressing large-scale vortices generated by the pump and tube bends. The outflow from the plenum is directed vertically and passes a nozzle with a contraction ratio of 7.4. A weir in the upper plenum maintains a constant water level and a free surface at atmospheric pressure. The water from the upper plenum returns to the reservoir passing a number of baffles giving time for air bubbles trapped in the water to rise to the surface. The test section has

Figure 2.2: Test loop

an asymmetric cross section corresponding to one quarter of the fuel bundle of SVEA – 96, illustrated in Fig.2.3. The same figure shows the spacer part in the studied subchannels. The 24 rods are held in position by 5 spacers. The spacers are of thin plate spring construction; the details of the spacer can be seen on the photographs in Fig.2.3. The characteristic geometric parameters of the test section
CHAPTER 2. FLOW AROUND SPACERS

Figure 2.3: The cross-sectional view of the test section. The subchannels where the velocity was measured are denoted by $A, B,$ and $C$. The subchannels where the pressure was measured are filled with grey color.

are given in Table 2.1. The walls of the test section are made of 10 mm thick glass plates, allowing visibility techniques to be used to study the flow.

2.2.2 Measurement technique

The pressure distribution inside the mock-up of the fuel bundle was measured by use of pressure probes. Five pressure taps were drilled in one of the rods, aligned along a vertical straight line. The pressure sensing rod could be rotated along its axis and traversed axially within a range of $\pm 250$ mm. The position of the holes and the traversing range enables to get a detailed vertical pressure distribution in the test section. The pressure taps are connected from the inside of the rod with plastic tubes to a set of differential pressure transducers, positioned at the level of the free surface in the upper plenum, thus compensating for the gravitational...
2.2. EXPERIMENTAL ARRANGEMENT

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rod diameter</td>
<td>10.0 mm</td>
</tr>
<tr>
<td>P/D (pitch-to-diameter ratio)</td>
<td>1.32</td>
</tr>
<tr>
<td>W/D (wall-to-diameter ratio)</td>
<td>1.305 and 1.264</td>
</tr>
<tr>
<td>Rod length</td>
<td>1894 mm</td>
</tr>
<tr>
<td>Cross-section flow area</td>
<td>2400 mm</td>
</tr>
<tr>
<td>Hydraulic diameter</td>
<td>9.86 mm</td>
</tr>
</tbody>
</table>

Table 2.1: Geometric characteristics of test section

pressure head in the vertical test section. The flexible plastic tubes are translucent to ensure proper purging of air bubbles prior to testing.

The desired arrangement of the pressure sensing rod was initially established at specific radial and axial positions, which involved dismantling of the test section when the radial position was changed. The next step was to reach steady pump rotations at the desired liquid flow rate. The accumulated air in the plastic tubes which connect the pressure taps with the transducer was properly purged. The traversing system of the pressure sensing rod was connected to the LABVIEW programme, where the desired translation distance and the step were entered before each run. The data of each run were automatically saved to the computer. The rate was 1000 samples per second and the sampling time was set to 10 s for each position. The waiting time before sampling was set to 10 s in order to avoid the influence of rod movement.

In order to have good spatial and temporal resolution and minimal probe interference with the local flow structure, Laser Doppler Velocimetry (LDV) technique was used to measure axial velocity and the turbulence intensity across the third spacer of the test section. Polytec LDV system used in the present work has been operated in dual-beam back-scatter mode. The 61.178 mm beam separation and 310 mm focal length focusing lens result in beam waist at the measurement volume equal to 68 µm. To avoid the influence of vibrations of the test loop on the measurement equipment, the LDV with its 3D traverse system was installed on a separate platform.

The start of the measurement run was the adjustment of the desired water flow rate. The initial position of the LDV measurement volume was selected and carefully placed starting from a reflecting surface, such as rod surface or box wall and translating the system by desired distance. Before each run a few test measurements were observed on the oscilloscope screen to make sure that the signal-to-noise ratio is appropriate, if not, it was adjusted to satisfy the requirement. The translation algorithm was introduced into the computer. The data were automatically saved.

2.2.3 Accuracy

The water flow was measured using ABB Kent-Taylor electromagnetic MagMasterTM flowmeter with stated measuring accuracy ±0.15%. Over the time period
required to obtain a full axial pressure profile or a full velocity run, the standard deviation of the fluctuating flowmeter readings was ±1% or less.

Calibrated Motorola MPX-5050 piezoresistive differential pressure transducer was used to sense the pressure. The calibration of the transducer showed a maximum error of 6%. The pressure tap locations were calibrated before each set of measurements against a reference point by means of a ruler. The maximum error in location reading was considered to be less than 0.5 mm.

During velocity and turbulence intensity measurements a reasonable signal-to-noise ratio was chosen by adjusting the shape of the probability density function curve to represent a Gaussian distribution by modifying the burst threshold. The raw signal together with the triggering of the digitized data were observed on the oscilloscope screen to make sure that the triggering occurs when the burst does. To have a reliable statistical value, at least 2000 valid counts were accumulated, which in some cases accounted for 20-30 min measuring time in one point. The biggest source of error was considered to be the initial positioning of the LDV measurement volume. The position was measured by a ruler and sight judgment accounting for 0.5 mm error in readings, which for the subchannel width of 3 mm is a significant source of error.

2.3 Experimental findings

The mobile pressure rod permitted to obtain a large database of pressure measurements in the rod bundle. The existing correlations for the pressure drop for bare rods and for the spacers were tested using the measurements. More on this information is found in Paper 1. The advantage of the mobile rod is the possibility to measure pressure inside the spacer, quantifying the effect of the geometrical details. An example of pressure development inside the spacer for two different subchannels is presented in Fig.2.4. The main characteristic is that there is a sudden pressure drop at both edges. Inside the spacer, the pressure is very sensitive to whether there is a spring or other details in each subchannel. This is especially valid for the internal subchannels. The near-wall subchannels exhibit less sensitivity. The detailed pressure information is valuable in validating the mechanistic models of fuel rod bundle flows.

With the LDV, the streamwise instantaneous velocities were measured. From these data, mean velocity values and the fluctuating components were extracted. The streamwise mean velocity for two subchannels is presented in Fig.2.5. A sudden increase of about 20–30% in the mean velocity is observed downstream of the spacer with a monotonic decrease with the distance from the spacer. The acceleration of the flow is mainly due to the contracted area. The closer-to-wall subchannel exhibits a slower relaxation to the upstream state than the internal subchannel.

The behaviour of the streamwise fluctuations, shown in Fig.2.6 for two subchannels, is different depending on the placement of the subchannel. Through this study, it is possible to confirm the observations of Rowe and Chapman (1973), by
2.3. EXPERIMENTAL FINDINGS

Figure 2.4: Detailed pressure measurement inside the spacer grid. The flow is from right to left.
CHAPTER 2. FLOW AROUND SPACERS

Figure 2.5: Streamwise mean velocity
2.3. EXPERIMENTAL FINDINGS

Figure 2.6: Streamwise component of fluctuating velocity

which the fluctuating velocity in wall subchannel increases at the trailing edge of
the spacer, followed by a monotonic decrease, whereas, for the internal subchannel
it dives below the upstream value with the subsequent large increase. The increase
just at the trailing edge of the spacer is not seen possibly to the limited visibility
of the LDV technique very close to the spacer.

For the wall subchannel the downstream increase is of $15\text{–}40\%$. The Nagayoshi&
Nishida formula in Eq.(2.1), describes relatively well this type of behaviour. Obviously, Eq.(2.1) is not very useful in describing the internal subchannel behaviour. Paper 1 describes the experimental findings in more detail.

As it was mentioned in Section 2.1 the author believes that the different behaviour in turbulence for the internal subchannels is due to a global velocity profile formed in the rod bundle. This global velocity profile then triggers the redistribution of turbulence energy. The formation of the global velocity profile was recently confirmed by Dominguez-Ontiveros et al. (2010). To confirm the redistribution of turbulence energy, an experimental study of all fluctuating components is needed.

2.4 Computational Fluid Dynamics predictions

Computational Fluid Dynamics calculations using CFX 10.0 were performed in order to compare to the experimental results. The geometry was limited to one subchannel at a time. Subchannels A, B, and C were modelled. The upstream and the downstream of the spacer parts constitute $>50D_h$. The mesh is unstructured throughout the whole geometry, but has been generated separately for the subchannel part and spacer part. It consists of tetrahedral elements and prismatic layers close to the wall. The total number of elements varied between 1,200,000 and 1,300,000. The $y^+$ value varied between 3-20 for different walls. The mesh of spacer part of subchannel B is illustrated in Fig.2.7.

The calculated pressure in subchannels A and B is presented in Fig.2.8. The overall pressure drop over the spacer is underpredicted by $20 - 30\%$. The calculated pressure distribution inside the spacer is distinct from the measured pressure. Subchannel A for example does not contain any springs. The rod is braced by two symmetrically arranged bumps at both ends of the spacer. Thus, the calculated pressure has a smooth distribution inside the spacer, except at the two edges. However, the measured pressure exhibits more fluctuations. These fluctuations can possibly be explained by the existing cross-flows between subchannels, which cannot be captured in a single subchannel model. In subchannel B, where there is a spring, the calculated pressure fluctuates. However, these fluctuations are different from the measurements.

The measured pressure distribution inside the spacer for both analysed subchannels, it is worth pointing out that they are neighbouring subchannels, has almost the same shape. Since geometrically they are quite different due to the presence of the spring in subchannel B, the cause must be an external source outside the subchannels, which has a strong influence on both.

Thus, in order to obtain the right pressure distribution inside a spacer it is not enough to model just a separate subchannel, even if the exact geometry is reproduced. It might be necessary to model the whole fuel bundle, since the influence of the cross flows is quite significant. For the prediction of pressure drop over the spacer it might be enough to have a working correlation to have the same precision as in the case of a single subchannel model with detailed geometry reproduction.
Figure 2.7: Mesh of the spacer part of subchannel B
CHAPTER 2. FLOW AROUND SPACERS

Figure 2.8: Comparison of calculated values of pressure to the experimental ones

Figure 2.9: Comparison of calculated values of mean streamwise velocity to the experimental ones
The calculated streamwise mean velocities are compared to the measured velocities in Fig.2.9. The values are scaled by the upstream velocities, in order to avoid to some extent the error in positioning the LDV measuring volume. Because there is a velocity profile the comparable values are quite sensitive to the positioning of the sampling probe in the subchannel cross-section. More on this topic can be read in Paper 2. As can be seen from the figure, the mean streamwise velocities are predicted well for the analysed region.

From CFD simulations, the RMS fluctuating component of the velocity, assuming turbulent isotropy is calculated as:

\[ v' = \sqrt{\frac{2}{3}} k \]  \hspace{1cm} (2.2)

where \( k \) is the turbulence kinetic energy.

The comparison of the calculation results to the measured RMS fluctuating velocities is presented in Fig.2.10. The discrepancy between the model calculations and the measurements is large and cannot be explained solely by the sensitivity in placing the sampling probe. A share of this discrepancy might be attributed to the imperfect match in the geometry, since every small angle and geometry detail will contribute to the turbulence generation or suppression. It is worth mentioning that very sharp angles, where the spacer details touch the rods were deliberately enlarged to avoid problems in meshing algorithms. Though, this is an important point to consider, it is suspected that the turbulence models, which were tested, cannot catch the wakes and vortices created by the complicated geometry of the spacers. The other possible reason for this discrepancy, which was discussed above, is that the single subchannel model does not take into consideration the global velocity profile in the rod bundle.
Figure 2.10: Comparison of calculated values of streamwise fluctuating velocity component to the experimental ones.
Chapter 3

Drop deposition. Influence of obstacle on deposition

3.1 Problem statement

Drop deposition in annular two-phase flow is still a scientific challenge. It has been shown, by Young and Leeming (1997) for example, that the mechanism of deposition depends on drop/particle inertia. They distinguish three mechanisms: Brownian diffusion for very light drops/particles, dispersion driven by turbulence of the continuous phase, and direct impaction for most massive drops/particles. In annular two-phase flow the drops have a distribution of sizes. Additionally, the drops coalesce or split into smaller drops in the course of the flow, changing their inertia and, thus, possibly, the deposition mechanism. A number of mechanistic models are proposed in the literature to calculate the drop deposition. These models can be grouped, by the formulation of the drop equations, into two main groups: Lagrangian (Kallio and Reeks, 1989; Narayanan et al., 2003) and Eulerian (Young and Leeming, 1997; Zaichik et al., 2008). Paper 3 discusses in more detail the two types of models.

The Lagrangian-type model, called Lagrangian Particle Tracking, is attractive in several ways for the prediction of drop deposition. Firstly, the boundary conditions of complete absorption, an elastic rebound or an intermediate state is easily implementable. All these boundary states are encountered in annular two-phase flow. Secondly, Lagrangian Particle Tracking permits, with little complexity, to include a drop size distribution, resembling the real flow of drops. Thirdly, the break-up or coalescence of drops is relatively simple to implement. One of the drawbacks of this type of model is that the two-phases do not share the same discretization, inducing interpolation errors. Additionally, a large number of representative drops have to be tracked through the flow in order to keep the statistical confidence.

In Eulerian description, the drop phase is treated as a continuum. This allows the disperse phase to be treated with the same discretization and the same numer-
CHAPTER 3. DROP DEPOSITION. INFLUENCE OF OBSTACLE ON DEPOSITION

ical scheme as the continuous phase, avoiding the interpolation errors. It becomes more important when the disperse phase exerts an influence on the gas. Distinct wall boundary conditions for the drops are more difficult to implement. In annular two-phase flow, the disperse phase is present with a range of sizes. If this range of sizes is distributed into \( n \) bins, then \( n \) sets of conservation equations for mass, momentum and eventually, energy are to be solved. This substantially increases the computational cost.

In Chapter 1 it was stated that a model for calculation of drop deposition around spacer is needed in order to avoid the empiricism in prediction of dryout. At the same time allowing the possibility for spacer geometry optimization. Several attempts have been made to use Lagrangian Particle Tracking (LPT) to predict drop deposition, in channels with obstacles resembling the flow in nuclear fuel assemblies. Windecker et al. (1999) use LPT in channels with varying obstacle geometry to obtain a correlation of deposition rate as a function of turbulent kinetic energy. The model is compared to one set of air-water measurements. Yamamoto et al. (1997) use LPT to analyze the drop deposition for several nuclear fuel assembly spacer designs. No direct comparison to experimental data is performed. The validation of the model is made by comparing the development trend of deposition and the dryout behaviour. Naitoh et al. (2002) employs LPT to calculate the increase in deposition downstream of the spacers in a nuclear fuel assembly. The results are incorporated into an integral code to predict the dryout. No direct comparison is made to check whether the prediction of drop deposition is correct. No model from the Eulerian group has been tested in geometries other than a pipe. Thus, there is still a need for careful testing of drop deposition models.

3.2 Phenomenological description

This Section describes the mechanisms of drop deposition and the parameters of influence. One of the major parameters of influence is the drop size. Thus, the formation of drops is analysed together with the ways to predict an average drop size.

3.2.1 Process of deposition

The drop deposition as an averaged process is viewed as driven by the drop concentration. This is how it is analysed and modelled. The deposition rate per unit area of wall \( \dot{m}_d \) is calculated as:

\[
\dot{m}_d = k_d C
\]

where \( k_d \), called deposition mass transfer coefficient or deposition velocity, is an experimentally correlated parameter; and \( C \) is the channel-averaged drop concentration.
3.2. PHENOMENOLOGICAL DESCRIPTION

The most cited correlation for the deposition mass transfer coefficient is the one of Govan et al. (1988). This correlation is relevant for applications in Boiling Water Reactor because the data base included high pressure steam-water flows. It is reproduced in Fig. 3.1. The group with which the deposition mass transfer coefficient is non-dimensionalised, $\sqrt{\rho g D_h/\sigma}$, is derived from the definition of the modified Weber number $W_{cm}$, which is:

$$W_{cm} = \frac{\rho_g j_g^2 D_h}{\sigma} \quad (3.2)$$

where $j_g$ is the superficial velocity of the gas, $D_h$ is the hydraulic diameter of the channel, $\rho_g$ is the density of the gas and $\sigma$ is the surface tension of the liquid. The entrainment of drops from a continuous surface of liquid starts when $W_{cm} \geq 1.0$. Thus, the so-called critical gas velocity, at which the entrainment starts is defined as follows:

Figure 3.1: Deposition mass transfer coefficient as a function of homogeneous concentration. Reproduced from Hewitt and Govan (1990)
CHAPTER 3. DROP DEPOSITION. INFLUENCE OF OBSTACLE ON DEPOSITION

Figure 3.2: Deposition as a function of particle inertia. Reproduction from Young and Leeming (1997)

\[ j_{g,cr} = \sqrt{\frac{\rho_g D_h}{\sigma}} \]  \hspace{1cm} (3.3)

This velocity \( j_{g,cr} \) is used to non-dimensionalize the deposition velocity \( k_d \). Thus, the correlation of Govan et al. (1988) is defined by the two most important parameters, \( j_{g,cr} \) - setting the limit for drop formation and the actual drop concentration, resulting in the successful application of the correlation.

If a mechanistic model of drop deposition is desired then the process of deposition has to be understood on a local basis. In general, the drop deposition is considered to be driven by turbulence dispersion in the core of the channel. There is, however, no consensus about how the drop traverses the quiescent flow region at the wall.

As it was discussed above, three deposition regimes as a function of drop inertia have been identified. An illustration is given in Fig. 3.2.

Herein, the notation for 'particle' and 'drop' will be interchanged to comply with the notation in the literature. This is because most of the analysis was made for deposition of solid particles, which is applicable to liquid drops as well. In Fig. 3.2, \( \tau_p^+ \) is the non-dimensional particle relaxation time. The particle inertia is defined by its relaxation time, which for a spherical particle in Stokes flow is calculated using Eq. (3.4).
3.2. PHENOMENOLOGICAL DESCRIPTION

\[ \tau_{p0} = \frac{\rho_p d_p^2}{18 \rho_f \nu_f} \quad (3.4) \]

where \( \rho_p \) is the material density of the particle, \( d_p \) is the diameter of the particle, \( \rho_f \) is the density of the fluid surrounding the particle and \( \nu_f \) is the kinematic viscosity of the fluid. The correction for higher Reynolds number flow takes the form of the Eq.(3.5).

\[ \tau_p = \frac{\tau_{p0}}{\varphi(\text{Re}_p)} \quad (3.5) \]

\[ \varphi(\text{Re}_p) = \begin{cases} 1 + 0.15 \text{Re}_p^{0.687} & \text{at } \text{Re}_p \leq 1000 \\ 0.11 \text{Re}_p/6 & \text{at } \text{Re}_p > 1000 \end{cases} \]

\[ \text{Re}_p = \frac{d_p |\mathbf{u} - \mathbf{v}_p|}{\nu_f} \]

where \( \mathbf{u} \) is the fluid velocity vector and \( \mathbf{v}_p \) is the particle velocity vector.

The particle relaxation time is usually non-dimensionalized by the viscous units of the continuous phase, as in Eq.(3.6)

\[ \tau_p^+ = \tau_p \frac{u_*^2}{\nu_f} \quad (3.6) \]

where, \( u_* \) is the friction velocity.

Three regions of deposition are delimited. For very light particles for which \( \tau_p^+ \ll 0.3 \), the transport to the wall is well represented by turbulent diffusion in the core of the channel and Brownian diffusion in a very thin layer adjacent to the wall. The Brownian diffusion is the motion of tiny particle due to random forces from the movements of the surrounding fluid molecules. This regime is usually termed **diffusional regime**. Here, the deposition rate is small and \( k_d^+ \) is nearly independent of \( \tau_p^+ \).

In the intermediate regime, called **diffusion-impaction regime**, \( 0.3 < \tau_p^+ \lesssim 20 \), the deposition mass transfer coefficient steeply increases with the increase in particle relaxation time. The particles in this regime move in the turbulent flow under the influence of gradient of turbulence. However, researchers still do not agree upon the theoretical basis on how the particles traverse the boundary layer. Two hypotheses are advanced. The first theory, initiated by Friedlander and Johnstone (1957) variously referred as 'free-flight' or 'stopping-distance' model. The essence of the model is that particles are transported by turbulent diffusion to within one 'stop-distance' of the wall where they acquire sufficient inertia to reach the wall. A recent application of the theory of 'free-flight' can be found in Marchioli et al. (2003). The second hypothesis was advanced by Cleaver and Yates (1975). They say that particles are convected to the wall if they are entrained by sweeps. The sweep is part of near-wall coherent structures, which are defined by Robinson (1991).
as a three-dimensional region of the flow over which at least one fundamental flow variable (velocity component, density, temperature, etc.) exhibits significant correlation with itself or with another variable over a range of space and/or time that is significantly larger than the smallest local scales of the flow. A recent application of the hypothesis of Cleaver and Yates (1975) to calculate particle deposition can be found in Guingo and Minier (2008).

The third regime, called inertia-moderated regime, is characterized with the slow decrease in deposition mass transfer coefficient as the particle relaxation time increases. In this regime the particles of $\tau_p^+ > 20$ acquire enough inertia from the core turbulence to cast themselves to the wall (liquid film). The slow decrease in deposition mass transfer coefficient is discussed to be due to less responsiveness to turbulence once the drop size increases. For this deposition regime, the initial conditions of drop formation might be the determining factors. Especially, this is true for drops in annular two-phase flow, which are formed by entrainment from the liquid film. James et al. (1980) in their photographic studies observed that large drops, detached from the wavy liquid film, travelled almost in straight lines to the opposite wall, unaffected by turbulence.

### 3.2.2 Drop size

One of the reasons, which makes drop dynamics analysis in annular two-phase flow particularly difficult, is that the drop size is not known a priori. The drops are continuously created from the wave crests, from the splashing of drops on the liquid film, from breaking-up of larger drops and from coalescence. The result of all these processes is a large range of drop sizes. In describing the distribution of drop sizes in annular two-phase flow it is convenient to work with the number distribution function $f_n$. It is defined in Eq. (3.7).

$$\frac{dN}{dd_p} = f_n(d_p)$$  
$$\int_0^\infty f_n(d_p)dd_p = 1$$

where $N$ represents the total number of drops and $d_p$ is the drop diameter. This gives the fraction of drops with diameters between $d_p$ and $d_p + dd_p$. Using the number distribution, varying types of drop diameter averages can be defined. Eq. (3.8) is the general relation to calculate these averages.

$$d_{mn} = \left( \frac{\int_0^\infty d_p^m f_n(d_p)dd_p}{\int_0^\infty d_p^n f_n(d_p)dd_p} \right)^{1-\frac{m}{n}}$$  

where usually the indices $m, n \in [0 : 4]$. It is also common to define a volume distribution function $f_v$, which is $dv/dd_p$. Some largely used diameter averages are described in Table 3.1.
3.2. PHENOMENOLOGICAL DESCRIPTION

<table>
<thead>
<tr>
<th>mean diameter</th>
<th>name</th>
<th>field of application</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d_{10}$</td>
<td>arithmetic or linear</td>
<td>evaporation</td>
</tr>
<tr>
<td>$d_{20}$</td>
<td>surface</td>
<td>surface area controlling (e.g. absorption)</td>
</tr>
<tr>
<td>$d_{30}$</td>
<td>volume</td>
<td>volume controlling (e.g. hydrology)</td>
</tr>
<tr>
<td>$d_{21}$</td>
<td>surface diameter</td>
<td>adsorption</td>
</tr>
<tr>
<td>$d_{31}$</td>
<td>volume diameter</td>
<td>evaporation, molecular diffusion</td>
</tr>
<tr>
<td>$d_{32}$</td>
<td>Sauter</td>
<td>efficiency studies, mass transfer</td>
</tr>
</tbody>
</table>

Table 3.1: Diameter averages

Figure 3.3: Example of number and volume based drop size distributions showing the position of the Sauter mean diameter and other important mean diameters. Reproduced from Thermopedia.

As indicated by Table 3.1, the Sauter mean diameter is the diameter suitable in application of mass transfer, and it is the most used average in drop deposition calculations. An example of relative positioning of different averages is illustrated in Fig. 3.3.

The mechanism of drop formation and the drop stability can be expressed in terms of a balance between external stresses, $\tau$, and the surface stress, $\sigma/(\rho g)$. Basically, there are two external stresses, which are involved in breaking-up of drops, namely, viscous and dynamic stresses. The relative order of magnitude of these stresses is determined by the ratio of length scales, $d_{\text{max}}/\lambda$, where $d_{\text{max}}$ is the maximum stable drop diameter, and $\lambda$ is the internal length scale of local turbulence.
CHAPTER 3. DROP DEPOSITION. INFLUENCE OF OBSTACLE ON DEPOSITION

(Kolmogorov microscale). When \( d_{\text{max}} \leq \lambda \) viscous forces play a dominant role. In most cases, including the annular two-phase flow in nuclear fuel assemblies, the viscous effects are negligible. Thus, the dynamic disruptive forces and stabilizing surface tension are dominant in the process of fluid break-up.

The disruptive forces may develop either through the local relative motion around the drop, or through the changes in eddy velocities over the length of a drop. Researches propose varying forms for the disruptive forces, see for example Kataoka et al. (1983) and Kocamustafaogullari et al. (1994). The purpose of this work has not been to analyse in detail the process of entrainment, but rather have a working relation in predicting the Sauter mean drop diameter as an input parameter to the deposition model.

In Paper 4 a thorough investigation is conducted on parameters of influence on the Sauter mean drop diameter. Several prediction correlations are tested against a large data basis of measured drop sizes. However, none of the tested correlations can predict with acceptable accuracy the drop size. In the course of the analysis of the parameters of influence, the main trends of several of these parameters have been identified. These are the gas Reynolds number based on superficial gas velocity, \( \text{Re}_g \), the modified Weber number, as in Eq. (3.2), the entrained liquid mass flux, \( G_{le} \). As a result a new correlation has been set, when has the form of Eq. (3.9).

\[
d_{32} = 0.0001D_h \left[ \left( \frac{\text{Re}_g}{W_e} \right)^{0.85} \left( \frac{\mu_g}{\mu_l} \right)^{0.15} + \left( \frac{G_{le}}{\rho_h \rho_g \rho_l} \frac{\mu_l}{\mu_g} \right)^{0.75} \right]^{0.05}
\]

(3.9)

The ratio of densities and viscosities is included in order to obtain a better tuning. This is because few measurements are available, which investigate the influence of these parameters. The comparison of the correlation prediction is illustrated in Fig. 3.4.

Even though all the data are correlated, the accuracy of the correlation is in the range of \( \pm 40\% \) in the range of compared data. Thus, it is suggested that a systematic experimental study is undertaken where all the parameters of influence are varied with a considerable range.

As to clarify which deposition regime can be attributed to the drops in annular two-phase flow, a large set of measured drop sizes is plotted in Fig. 3.5.

If the Sauter mean diameter can be considered the characteristic diameter, the drops in annular two-phase flow should follow the inertia moderated regime. On the one hand there is no confusion about how the drops traverse the quiescent region of the flow close to the wall. On the other hand, the deposition model should ensure that the scales (time and length) differ from the scales of the fluid flow. The problem of specifying the drop initial conditions, e.g. drop size and initial velocity, is more important for these massive drops.
3.2. PHENOMENOLOGICAL DESCRIPTION

![Figure 3.4: Comparison to eq.(3.9)](image1)

![Figure 3.5: Inertia characteristic of drops in annular two-phase flow](image2)
CHAPTER 3. DROP DEPOSITION. INFLUENCE OF OBSTACLE ON DEPOSITION

3.3 Lagrangian Particle Tracking

The Lagrangian-type models to calculate drop deposition, in general, should provide a more detailed and realistic model of deposition because the instantaneous equations of motion are solved for each representative particle through a field of random fluid eddies. Hutchinson et al. (1971) were the first to introduce a stochastic element into the theory of deposition. Other models followed. Here, the model of Gosman and Ioannides (1983) is used. In this model, the particle is always assumed to be within a single eddy. Each eddy has a characteristic fluctuating velocity, \( u_f \), length, \( l_e \), and lifetime, \( \tau_e \). The continuous phase flow is calculated with Reynolds Averaged Navier-Stokes (RANS) equations. The turbulence is modelled by \( k - \epsilon \) equations. Thus, the averaged velocity components, the turbulence kinetic energy, \( k \), and the dissipation of the turbulence energy, \( \epsilon \), are known values from the solution. The eddy characteristics are calculated as in Eq. (3.10).

\[
\begin{align*}
u_f &= \Gamma \sqrt{\frac{2}{3}} k \\
l_e &= \frac{C_\mu^{3/4} k^{3/2}}{\epsilon} \\
\tau_e &= \frac{l_e}{\sqrt{\frac{2}{3} k}}
\end{align*}
\]  

where \( \Gamma \) is a normally distributed random number, for which the mean is zero and the standard deviation is equal to one; and \( C_\mu \) is the turbulence constant, here equal to 0.9.

When a particle enters an eddy, the fluctuating velocity for that eddy is added to the local mean fluid velocity to obtain the instantaneous velocity. The turbulent fluid velocity, \( u_f \), is assumed to prevail as long as the particle-eddy interaction time is less than the eddy lifetime and the displacement of the particle relative to the eddy is less than the eddy length. If either of these conditions is exceeded the particle is assumed to be entering a new eddy, with new characteristic fluctuating velocity, length and lifetime. The interaction time between the eddy and the drop \( \tau_{int} \) is a minimum between the eddy lifetime and the transit time. The transit time is estimated from the following solution of a simplified and linearized form of the equation of motion of the drop:

\[
\tau_{tr} = -\tau_p \ln \left( 1.0 - \frac{l_e}{\tau_p |u_f - u_p|} \right)
\]  

If \( l_e > \tau_p |u_f - u_p| \), Eq.(3.11) has no solution. This is interpreted as the capturing of the particle by the turbulent eddy, in which case \( \tau_{int} = \tau_e \).

Particle \( i \)'s position and velocity are calculated as in Eq. (3.12) and Eq. (3.13), respectively.
3.3. LAGRANGIAN PARTICLE TRACKING

Figure 3.6: Influence of gas Reynolds number on deposition. For all cases the concentration of drops was kept constant and equal to $C = 0.1 \, [kg/m^3]$

\[ x_i^n = x_i^o + u_{pi}^o \delta t \]  \hspace{1cm} (3.12)

\[ u_{pi}^n = u_f + (u_{pi}^o - u_f) e^{-d_p/\tau_p} + \tau_p F_{all} \left(1 - e^{-d_p/\tau_p}\right) \]  \hspace{1cm} (3.13)

where superscripts $n$ and $o$, denote new position and old, respectively; $\tau_p$ is the particle relaxation time, calculated with Eq. (3.5); $F_{all}$ incorporate all the forces, which act on the particle. Because $\rho_l \gg \rho_g$ only drag and gravity forces were taken into consideration.

A synthetic case of vertical pipe flow was set to study the calculation trends. Two parameters were systematically varied: gas Reynolds number, $Re_g$, and drop concentration, $C$. The drop diameter, $d_p$ was calculated with the correlation Eq. (3.9). Calculations were also performed with drop diameter corresponding to the accuracy limits of the correlation, that is $d_{32} + 40\%$ and $d_{32} - 40\%$.

Fig.3.6 illustrates the influence of the gas Reynolds number on the deposition velocity (deposition mass transfer coefficient). The deposition mass transfer coefficient increases with increasing gas turbulence. This trend is noticeable in some experiments of deposition, for example for helium-water of Jepson et al. (1989). However, the correlations, which are used to calculate the deposition mass transfer coefficient (Govan et al. (1988)) do not contain the effect of gas Reynolds number. Notice that smaller drop sizes result in larger deposition mass transfer coefficient. It can be also noticed that an underestimation of drop size results in a larger error compared to the case of overestimation.
CHAPTER 3. DROP DEPOSITION. INFLUENCE OF OBSTACLE ON DEPOSITION

Fig. 3.7 shows the model response to the variation in drop concentration. It is observed that the deposition mass transfer coefficient decreases with the increase in concentration. This trend is shown both in experiments (Jepson et al. (1989)) and by the prediction correlation (Govan et al. (1988)). The explanation for the decrease in deposition mass transfer coefficient with the increase in concentration is that at higher concentrations, the probability of coalescence rises. This results in larger drops, which respond more reluctantly to turbulence. De facto, there is no explicit term, which accounts for the effect of concentration in the movement of drops in our model. However, the drop size correlation in Eq. (3.9), takes into account the fact that the drop becomes larger with the increase in the entrained liquid mass flux, which corresponds to the true physical process.

The response of the model to the variation of the characteristic drop inertia parameter is illustrated in Fig. 3.8. The behaviour is correct considering that the inertia moderated regime is prevailing.

Compared to experiments, for which the drop mean Sauter diameter is measured, the model predicts the deposition mass transfer coefficient with ±30% accuracy. The comparison for data of Jepson et al. (1989) is illustrated in Fig. 3.9.

In the end, a model capable to predict the deposition in flows resembling the conditions of BWR is required. Hewitt et al. (1969) measured deposition rates in steam-water flows at 70 bar pressures with mass fluxes similar to fluxes in BWR.

Unfortunately, there is no knowledge about drop sizes in this kind of flows, especially because they are characterised with very high drop concentration, which is about three order of magnitude higher than the commonly studied flows. Another
3.3. LAGRANGIAN PARTICLE TRACKING

Figure 3.8: Influence of non-dimensional relaxation particle time on deposition

Figure 3.9: Measured versus calculated deposition velocity for air-water and helium-water cases from Jepson et al. (1989).
CHAPTER 3. DROP DEPOSITION. INFLUENCE OF OBSTACLE ON DEPOSITION

Figure 3.10: Deposition velocity for steam-water Hewitt et al. (1969)

A distinct characteristic is that the gas Reynolds number is at least one order of magnitude higher. Taking into consideration that Lagrangian Particle Tracking shows the trend of increasing deposition with the increase in gas Reynolds number, which is not observed in all experiments, this might need a further investigation.

The comparison of the model results with Hewitt et al. (1969) measured data is presented in Fig. 3.10. The discrepancy is quite big. If the drop size is tuned to match the experimental deposition data, drop of several millimeters are obtained. The details are presented in Paper 4.

Three reasons can be named for the discrepancy between the LPT model calculations and the experimental data of Hewitt et al. (1969):

1. the drop concentration is high, which might induce processes not taken into consideration in the model, like drop collision, coalescence and decrease of turbulence of the continuous phase

2. the LPT trend to calculate higher deposition with increased gas Reynolds number might not correspond to reality

3. the drop size is not known. As it was suggested above a systematic experimental study is needed to build a more solid knowledge about this input parameter to the deposition models. The drop size is directly connected to the high concentration.

3.4 Eulerian approach

In Eulerian framework the phases are defined as interpenetrating continua. From this concept averaged equations can be derived. The derivation and the form of equation can be found for example in Drew and Passman (1998) and Ishii and Hibiki (2011). Here, the attention is directed to the form of the mass source/sink, which is needed to define the deposition source/sink.
3.4. EULERIAN APPROACH

If there are two phases with an interchange of mass (liquid film and drops), the continuity equation can be written as follows:

\[
\frac{\partial \alpha_k \rho_k}{\partial t} + \nabla \cdot (\alpha_k \rho_k \mathbf{v}_k) = \Gamma_k \quad k = 1, 2
\]  \hspace{1cm} (3.14)

where \( \alpha_k \) is the volume fraction of phase \( k \), \( \rho_k \) is the material density of phase \( k \), \( \mathbf{v}_k \) is the velocity vector of phase \( k \) and \( \Gamma_k \) is the mass source/sink of phase \( k \).

The mass conservation at the interface yields:

\[
\sum_{k=1}^{2} \Gamma_k = 0
\]  \hspace{1cm} (3.15)

The mass source \( \Gamma_k \) is acting through an interphase, therefore a geometrical parameter needs to be defined. This parameter is termed interfacial area concentration \( a_i \), which quantifies the surface area of the interface per unit volume. Thus, the mass source can be defined as the product of an averaged mass flux through the interphase per unit volume. This is expressed in Eq. (3.16).

\[
\Gamma_k = \bar{\dot{m}}_k a_i
\]  \hspace{1cm} (3.16)

For a process involving only molecular diffusion the assumption of perfect absorption at the wall implies a zero concentration at the wall. However, this is not true for drops for which the motion length scale can be large compared to the scale characterising the variation of drop concentration. As a consequence the drop concentration has a finite value at the wall. Binder and Hanratty (1991) use the Razi Naqvi et al. (1982) radiation boundary condition to describe the particle mass flux at the wall. It is presented in Eq. (3.17).

\[
-\epsilon_p \frac{\partial C}{\partial r} \bigg|_{r=R} = \phi \sqrt{\frac{2}{\pi}} v'_p C \bigg|_{r=R}
\]  \hspace{1cm} (3.17)

where, \( v'_p \) is the root-mean-square of particle fluctuating velocity, \( \epsilon_p \) is the particle diffusivity defined as \( \epsilon_p = v'_p^2 / \tau_{pL} \), with \( \tau_{pL} \) being the Lagrangian time scale of the particle. In Eq.(3.17), \( C \) is the concentration of drops and \( \phi \) is the fraction of drops which are moving toward the wall. Binder and Hanratty (1991) took \( \phi \) to be equal to 1/2. The left-hand-side of Eq.(3.17) represents the deposition mass rate, which is a function of particle fluctuating velocity and the near wall concentration.

In this terms, the drop deposition rate per unit interfacial area of the liquid film can be defined as follows:

\[
\dot{m}_d = \phi \sqrt{\frac{2}{\pi}} \alpha_d \rho_d v'_d
\]  \hspace{1cm} (3.18)

The behaviour of drops in turbulent flow is governed by their interactions with turbulent eddies of the gas phase. Therefore, any statistical description of drops is dependent on the integral scales of the velocities of the fluid along the drops path.
CHAPTER 3. DROP DEPOSITION. INFLUENCE OF OBSTACLE ON DEPOSITION

These integral scales coincide with the corresponding Lagrangian integral scales for fluid particles in the limiting case of inertialess drops, that is, at \( \tau_p \to 0 \). In the case of highly inertial drops, \( \tau_p \to \infty \), the integral scales along drop paths should coincide with the corresponding Eulerian integral scales in the fluid, which express the statistical connection between fluctuations of parameters at fixed spatial points.

In other words, small drops with approximately the same density as the fluid will tend to follow the fluid so that \( v_d \approx v_f \). However, the more massive drops do not respond readily to the fluid fluctuations, and have a smaller tendency to change direction than the fluid, so that \( v_d < v_f \). The decrease in mean square velocity fluctuations has been shown to be proportional to the relaxation time of the particle. Tchen (Hinze (1975)) showed that the ratio fluid to particle velocity fluctuations is proportional to the ratio of the time scales as in Eq.(3.19).

\[
\frac{v_f^2}{v_d^2} = 1.0 + \frac{\tau_p}{T_L}
\]  

(3.19)

Here, \( v_f^2 \) is the square RMS fluctuating velocity of the fluid and \( v_d^2 \) is the squared RMS fluctuating velocity of the drop (particle).

Ideally, \( T_L \) should represent the Lagrangian integral time scale of the fluid following the path of an inertial drop, taking into account the crossing trajectories effect described by Csanady (1963). However, there is no general expression to calculate this quantity. This is why several expressions for \( T_L \) are tested.

Vames and Hanratty (1988) measured the turbulent dispersion of inertial drops in turbulent pipe flow. They have shown that the fluid integral time scale can be evaluated by Eq.(3.20).

\[
\frac{T_L u_*}{2R} = 0.046
\]  

(3.20)

Zaichik et al. (2008) pointed out that in isotropic turbulence, the Lagrangian integral time scale may be expressed in terms of local kinetic energy, its dissipation rate and the Kolmogorov constant. The constant depends on Reynolds number but assumes a constant value when Reynolds number is large. Zaichik et al. (2008) compiled a number of DNS data to find an optimal relation for the Kolmogorov constant \( C_0 \). The relation is presented in Eq.(3.21).

\[
T_L = \frac{4}{3} \frac{k}{C_0 \epsilon}
\]

(3.21)

\[
C_0 = \frac{C_{0\infty} Re_\lambda}{Re_\lambda + C_1}
\]

\[
Re_\lambda = \sqrt{\frac{15v_f^2}{\epsilon \nu}}
\]

\[
C_{0\infty} = 7.0
\]

\[
C_1 = 32.0
\]
3.4. EULERIAN APPROACH

where \( k \) is the turbulent kinetic energy and \( \epsilon \) is the dissipation of the turbulence. *Inertia moderated regime* presupposes massive drops, thus the "crossing trajectory" effect discussed by Csanady (1963) might have an influence. The "crossing trajectory" effect gains in importance when the condition in Eq. (3.22) is satisfied:

\[
\xi = \frac{|U - V|}{v'_{f,z}} > 1.0
\]  (3.22)

where \( v'_{f,z} \) is the RMS fluctuating velocity in the direction of gravity.

Csanady (1963) proposed expressions to calculate the integral time scale of the fluid on the path of a massive drop as follows:

\[
T_{L\parallel} = \frac{T_L}{\sqrt{1 + \beta^2 \xi^2}}
\]

\[
T_{L\perp} = \frac{T_L}{\sqrt{1 + 4\beta^2 \xi^2}}
\]

\[
\beta = \frac{T_L}{T_E}
\]

The subscripts (\([\parallel]\) and \([\perp]\)) denote the direction to the external force (gravity) as parallel and perpendicular, respectively. In this work, \( T_{L\perp} \) is used, because the normal-to-wall drop fluctuating velocity is needed, which is in the direction perpendicular to the gravity force. \( T_L \) here is calculated using Eq.(3.21) and \( \beta = \frac{3+2m}{3(1+m)} \), where \( m \) is the turbulence structure parameter. In this work, \( m = 1.0 \) is assumed. See the discussion about the magnitude of this parameter in for example Zaichik et al. (2008).

If the drop inertia is taken into consideration when evaluating the Lagrangian integral time scale of the fluid along the drop trajectory, this time scale may be considered as the interaction time between the drop and the turbulent eddy. Zaichik et al. (2008) proposed Eq.(3.24) to calculate the particle-eddy interaction without taking into consideration the average drift,

\[
T_{L_p}^{no\, drift} = T_L + (T_E - T_L)f(St_E)
\]

\[
f(St_E) = \frac{St_E}{1 + ST_E} - \frac{0.9mSt_E^2}{(1 + St_E)^2(2 + St_E)}
\]

where \( St_E \) is the Stokes number, calculated as \( St_E = \frac{T_L}{T_E} \).

Zaichik et al. (2008) proposed also a relation for the particle-eddy interaction taking into consideration the drift. Eq.(3.25) presents this relation.

\[
T_{L_p}^{drift} = \frac{2(1 + m_T^2 \xi^2)^{0.5} - m_T \xi}{2(1 + m_T^2 \xi^2)} T_{L_p}
\]

\[
m_T = \frac{T_{L_p}}{T_E} m
\]
In order to test this method of calculating the drop deposition rate, two-fluid calculations for gas and liquid drops were performed. These calculations were set in ANSYS CFX 13.0. The details can be found in Paper 5. The output from the two-fluid calculations included velocity distribution and turbulence parameters of the gas phase; and velocity and volume fraction distributions of the drop phase. A stand-alone programme was established to calculate the deposition rate. In this programme, the drop RMS fluctuating velocities were calculated according to Eq. (3.19), where \( T_L \) takes the various definitions discussed above. In this way all the needed variables: the drop volume fraction from the two-fluid calculations and the drop RMS fluctuating velocity resulted from the stand-alone programme, are there to calculate the deposition rate. Due to the lack of suitable turbulence model for the inertial drops, which is replaced by the inclusion of the turbulence dispersion force, the imperfection of the set-up is recognised. That is why two ways are devised to calculate deposition rate:

1. where the variables are taken from the cell close to the wall
2. channel-averaged variables are used

An example of comparison to measurements is illustrated in Fig. 3.11.

The deposition calculated from one cell close to the wall, even though all the drops are assumed to deposit (\( \phi = 1.0 \)), is underpredicted. This fact can be explained by the phenomenon discussed previously, that the drop scales are larger than the fluid scales, which is not taken into account in the two-fluid model. In
3.4. EULERIAN APPROACH

contrast, the channel-averaged approach shows good prediction, where most of the points are placed around the line where $\phi = 0.5$.

For the flow conditions resembling the Boiling Water Reactor flows a worse prediction is noticeable. As it is discussed in Section 3.3 these flows differ by a number of characteristics from the commonly studied flows. The free parameter $\phi$, which denotes the drops moving towards the wall, can be determined as in Eq. (3.26). It is seen in Fig. 3.12 that it exponentially correlates with drop concentration.

$$\phi = \frac{\dot{m}_d^{exp}}{\dot{m}_d^{calc}} \quad (3.26)$$

Or, it can be viewed in a different way. With the increase in concentration the drops usually coalesce or collide, which results in diminishing of fluctuating velocities. The fluctuating velocity, as defined in Eq.(3.18) directly affect the calculated deposition rate. In the future work, the increase in the drop size and the effect of collisions have to be incorporated into the model.

If $\phi$ is calculated from the exponential relation, the selected measurements can be predicted with $\pm 30\%$ accuracy. It is illustrated in Fig. 3.13.
CHAPTER 3. DROP DEPOSITION. INFLUENCE OF OBSTACLE ON DEPOSITION

3.5 Influence of obstacle on deposition

Several researchers (Okawa et al., 2011; Damsohn, 2011) observed in experiments a sudden increase in deposition downstream of an obstacle. This increase is not captured by the correlations and models presently used to calculate the deposition. That is why special empirical coefficients are derived for each spacer type in nuclear fuel assembly to quantify the effect of deposition increase. Both the Lagrangian Particle Tracking in Paper 4 and the Eulerian-type model in Paper 5 are tested in their capacity to predict the effect of flow obstacle. For comparison, the measurements of Okawa et al. (2011) are chosen, because they measured diabatic steam-water flows at 5, 10 and 15 bar pressure. The input parameter - drop size, was unfortunately not measured.

The model results calculated with Lagrangian Particle Tracking are compared with the measurements in Fig. 3.14. The correlation of Govan et al. (1988) is included in the comparison.

The comparison of deposition results from the Eulerian-type model to the experimental data is illustrated in Fig. 3.15.

Both type of models have the capability to predict the deposition increase due to obstacle. The recommendation to improve the models encompasses a better way to predict the drop size and the inclusion of the effect of increasing concentration.

Figure 3.13: Comparison of calculated data to measurements for all sets considered, \( \phi \) for all cases is calculated with exponential relation

\[
\begin{align*}
\text{measured deposition rate, [kg/m}^2\text{s]} & \\
\text{calculated deposition rate, [kg/m}^2\text{s]} & \\
\text{steam–water, 70 bar} & \\
\text{air–water, 1.5 bar} & \\
\text{helium–water, 1.5 bar} & \\
- 30\% & \\
+ 30\% & \\
\end{align*}
\]
3.5. INFLUENCE OF OBSTACLE ON DEPOSITION

Figure 3.14: Comparison of calculation results of Lagrangian Particle Tracking model to measured data of Okawa et al. (2011) and Govan et al. (1988) correlation

Figure 3.15: Comparison of calculation results of Eulerian model to measured data of Okawa et al. (2011) and Govan et al. (1988) correlation. For all cases $\phi = 0.5$. 
Chapter 4

Concluding remarks

The thesis has discussed ways to improve the flow modelling in nuclear fuel assemblies of Boiling Water Reactors striving for a more mechanistic approach. Nuclear fuel assemblies are of a complex geometry and effective engineering methods have to be developed to predict the phenomena. Optimization of the spacer alone can improve the thermal limit parameter, thus, improving the safety of the reactor and the economic competitiveness. Computational Fluid Dynamics techniques are promising tools to develop a mechanistic model of nuclear fuel assemblies. These models need to be carefully validated against local measured flow parameters before their use in operation or design. In this work, measurement techniques, such as mobile pressure rod and Laser Doppler Velocimetry are developed and tested in providing local data for validation of mechanistic models. Single subchannel CFD model testing against the measured data reveal the need to take into consideration the cross-flows, or eventually to model the entire rod bundle.

Dryout is the major operational and design parameter of Boiling Water Reactors. In the prediction of dryout, the drop deposition is calculated nowadays entirely with empirical correlations. The spacer effect on deposition is quantified by empirical coefficients, derived for each spacer type. To improve the understanding, a phenomenological description of the process of deposition is presented in the thesis. From this study it is deduced that drop size, turbulence of the continuous field and drop concentration are the crucial parameters, which determine the mechanism of deposition. Since the drop size is one of the input parameters to the deposition model, its prediction methods are analysed. As a result of this analysis a new correlation is proposed. If the drop size is correctly estimated, it was shown that Lagrangian Particle Tracking is capable of predicting the deposition increase downstream of the obstacle with ±30% accuracy. A new Eulerian-type model to calculate the inertial drop deposition is proposed. The main purpose to develop the deposition model was to incorporate the capacity to quantify the effect of flow obstruction. The model is based on the available data obtained from two-fluid calculations, as drop volume fraction, slip ratio and turbulence quantities of the
CHAPTER 4. CONCLUDING REMARKS

continuous phase. The calculated deposition is within $\pm 30\%$ of the experimental data for cases with relatively low drop concentration. The very high drop concentration flow of steam-water at 70 bar pressure indicated that further studies are needed to include the effect of increasing concentration. The capability of the model to predict the deposition increase downstream of the obstacle has been tested in a pipe flow with cylindrical obstacle for steam-water flows at three different pressures. The comparison of the calculated deposition rate to the measured deposition rate upstream and downstream of the obstacle shows that the model has the capability to capture the obstacle effect.


