On stability of solutions to the two-fluid models for dispersed two-phase flow

Reynir Levi Gudmundsson

TRITA-NA-0222

Licentiate’s Thesis
Royal Institute of Technology
Department of Numerical Analysis and Computing Science
On stability of solutions to the two-fluid models for dispersed two-phase flow

Reynir Levi Gudmundsson

Akademisk avhandling som med tillstånd av Kungl Tekniska Högskolan
framlägges till offentlig granskning för avläggande av teknisk licentiat-examen
fredagen den 29 november 2002 kl 10.00 i sal E3, Osquarsbacke 14, Kungl
Tekniska Högskolan, Stockholm.

ISBN 91-7283-392-0, TRITA-NA-0222, ISSN 0348-2952,
ISRN KTH/NA/R-02/22-SE,

© Reynir Levi Gudmundsson, November 2002
Royal Institute of Technology, Stockholm 2002
Akademisk avhandling som med tillstånd av Kungl Tekniska Högskolan framlägges till offentlig granskning för avläggande av teknisk licentiat-examen fredagen den 29 november 2002 kl 10.00 i sal E3, Osquarsbacke 14, Kungl Tekniska Högskolan, Stockholm.

ISBN 91-7283-392-0
TRITA-NA-0222
ISSN 0348-2952
ISRN KTH/NA/R-02/22-SE

© Reynir Levi Gudmundsson, November 2002

KTH, Stockholm 2002
Abstract

In this thesis the two-fluid (Eulerian/Eulerian) formulation for dispersed two-phase flow is considered. The inviscid formulation in one space dimension is known to be conditionally well-posed. On the other hand the viscous formulation is locally in time well-posed for smooth initial data, but with a medium to high wave number instability. In this study we consider two issues.

First, we study the long time behavior of the viscous model in one space dimension, where we rely on numerical experiments. We try to regularize the problem in a standard way. The simulations suggest that it is not possible to regularize in a standard way.

Secondly, we analyze the inviscid formulation in two space dimensions. Similar condition for well-posedness is obtained as for the inviscid formulation in one space dimension.

ISBN 91-7283-392-0 • TRITA-NA-0222 • ISSN 0348-2952 • ISRN KTH/NA/R-02/22-SE
I wish to thank my advisor, Dr. Jacob Yström, for all his support, guidance and encouragement throughout this work.

I would also like to thank all my friends and colleagues at NADA for making NADA a nice place to work at.

Financial support from the Swedish Foundation for Strategic Research (SSF) through their Multiphase Flow Program is gratefully acknowledged.
Chapter 1

Introduction to two-phase flow

Multiphase flow is a quite common phenomenon, it occurs both in nature and in technology. One of the most trivial example in nature is that clouds are droplets of liquid moving in gas. There are also numerous examples where multiphase flow occurs in industrial applications, for examples energy conversion, paper manufacturing, food manufacturing and medical applications. Due to the large number of applications where multiphase flow occurs, it is important to have accurate models.

Multiphase flow can give rise to very complex combinations of phases as well as flow structures. The simplest case of multiphase flow is two-phase flow. Under standard condition there are only four states of matters or phases, that is gas, liquid, solid and plasma phase. In most cases, and in this study, a mixture of two of the three first phases is only considered.

In two-phase flow, complicated interaction between the two phases can occur, for example boiling of water, melting of ice, solidification of metals and many other situations where mass transfer occurs between the phases. The rate of mass transfer can be hard to model. Reaction or mixing of phases can also lead to more complexity of the flow, for example it is very hard to track salt particles in water, the salt resolves (or diffuses) into the liquid phase. In this study we only consider two-phase flow for immiscible phases and where there is no mass transfer.

There are numerous classification methods in the literature for two-phase flow problems, due to the variety of these. A general classification, by Ishii [16], is to divide two-phase flow into four groups depending on the mixtures of phases in the flow. The four groups are, the flow of gas-liquid, gas-solid, liquid-solid and immiscible liquid-liquid mixtures. The last case is technically not a two-phase mixture, it is rather a single phase two-component flow, but for all practical purpose it can be considered as a two-phase mixture.
Chapter 1. Introduction to two-phase flow

Ishii does also another classification in his work, that is based on the interfacial structure and topological distribution of the flow structure. This classification is more difficult, since the structure of the flow can change continuously. Two-phase flow is here classified according to the geometry of the interface into three classes; separated flows, transitional or mixed flows and dispersed flows, see Figure 1.1. Note that each of these classes have subclasses defined by typical flow regimes.

This classification is not unique and there are number of other classifications for two-phase flows, see for example [15]. The two-phase flow phenomena appears in numerous industrial application so other classifications are often depending on combinations of the two phases as well on the flow structure. These classifications can be for a specific flow problem, for example in [19] is a classification for fluidization, that is flow of solid particles in gas or liquid, which is quite common in many industrial applications.

<table>
<thead>
<tr>
<th>Class</th>
<th>Typical regimes</th>
<th>Geometry</th>
<th>Configuration</th>
<th>Examples</th>
</tr>
</thead>
<tbody>
<tr>
<td>Separated flows</td>
<td>Film flow</td>
<td><img src="image" alt="Film flow" /></td>
<td>Liquid film in gas</td>
<td>Film cooling</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Gas film in liquid</td>
<td>Film boiling</td>
</tr>
<tr>
<td></td>
<td>Annular flow</td>
<td><img src="image" alt="Annular flow" /></td>
<td>Liquid core and gas film</td>
<td>Film boiling</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Gas core and liquid film</td>
<td>Condensors</td>
</tr>
<tr>
<td></td>
<td>Jet flow</td>
<td><img src="image" alt="Jet flow" /></td>
<td>Liquid jet in gas</td>
<td>Atomization</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Gas jet in liquid</td>
<td>Jet condensor</td>
</tr>
<tr>
<td>Mixed or transitional flows</td>
<td>Slug or plug flow</td>
<td><img src="image" alt="Slug or plug flow" /></td>
<td>Gas pocket in liquid</td>
<td>Sodium boiling in forced convection</td>
</tr>
<tr>
<td></td>
<td>Bubbly annular flow</td>
<td><img src="image" alt="Bubbly annular flow" /></td>
<td>Gas bubbles in liquid</td>
<td>Evaporators with wall nucleation</td>
</tr>
<tr>
<td></td>
<td>Droplet annular flow</td>
<td><img src="image" alt="Droplet annular flow" /></td>
<td>Gas core with droplets and liquid film</td>
<td>Steam generator</td>
</tr>
<tr>
<td></td>
<td>Bubbly droplet annular flow</td>
<td><img src="image" alt="Bubbly droplet annular flow" /></td>
<td>Gas core with droplets and liquid film with gas bubbles</td>
<td>Boiling nuclear reactor channel</td>
</tr>
<tr>
<td>Dispersed flows</td>
<td>Bubbly flow</td>
<td><img src="image" alt="Bubbly flow" /></td>
<td>Gas bubbles in liquid</td>
<td>Chemical reactors</td>
</tr>
<tr>
<td></td>
<td>Droplet flow</td>
<td><img src="image" alt="Droplet flow" /></td>
<td>Liquid droplets in gas</td>
<td>Spray cooling</td>
</tr>
<tr>
<td></td>
<td>Particulate flow</td>
<td><img src="image" alt="Particulate flow" /></td>
<td>Solid particles in gas or liquid</td>
<td>Transportation of wheat</td>
</tr>
</tbody>
</table>

Figure 1.1. Different regimes for two-phase flows according to Ishii [16], taken from [7].
1.1 Models of two-phase flows

When modeling two-phase flow, it is necessary to know what phenomena, effects and flow structures are important. The flow structure is often an important factor. As mentioned above, and illustrated in Figure 1.1, there are three classes of two-phase flow structures. The flow structure differs significantly between these classes, although the mixed or transitional flow is a combination of the other two classes.

Modeling separated and dispersed flows are very different tasks. In separated flow, the interface between the two phases is quite important and significant. The interface between the phases is of course important also for dispersed flow, but its structure and position is in general hard to track. In some cases the exact structure or position is of no importance, only some kind of average quantity is needed for fluid flow or transport analysis.

Models for two-phase flow can be categorized into two different groups. In the first group, there are models that track the interface between the two phases. These are ideal models for separated flows. In the second group, there are models where the exact position of the interface is not followed specifically. Dispersed flows are usually modeled using models from this second group.

Ideally, one would like to track the interface between the phases at all time, for both the separated and the dispersed flows, similar to resolving all relevant scales of turbulent single phase flow (DNS). This is often too computationally expensive and sometimes redundant. In many cases, specially for dispersed flow, a bulk model is what is sought.

1.1.1 Interface tracking models

For separated two-phase flow, where the structure of the interface between the two phases is not complexed, it should be possible to use a model for the established single phase flow equations and moving boundary between the phases. However, many flows are too mixed for this method to be usable.

There exist a couple of interface tracking methods that have been applied to two-phase flow problems. In interface tracking methods, the interface is represented by a continuously updated discretization. These discretizations can be of Lagrangian or Eulerian type, depending on the method used.

One early method is the Marker and Cell (MAC) method [14]. There a number of discrete Lagrangian particles are advected by the local flow and the distribution of these particles identifies the regions occupied by a certain phase. There are a couple of other methods based on similar principle, for example a front-tracking method introduced in [29]. That method was designed to simulate the motion of bubbles in a surrounding fluid, but it has been used to simulate other phenomena concerning interface tracking. Those are only some examples of existing methods; a review of similar methods, along with a comparison of them, is presented in [27].

Behind the level-set method, introduced in [24], is a different idea compared with particle methods like the MAC method. In the level-set method, which is
based on a fixed Eulerian grid, the interfaces are defined as the zero level set of a continuous function that is updated in order to follow the position of the interface.

Each of these methods has its positive sides and its negative sides. The particle methods, like the front tracking method, are easy to implement and the interface force can be applied accurately, but they are usually computationally expensive. On the other hand using the level–set method it is easy to reconstruct the interface, but this method is not mass conserving in general.

1.1.2 Models for dispersed flows

The complexity of the interfaces between the two phases, in dispersed flow, is too high for that interface tracking methods to be suitable, at least with today’s computing capacity. To model this type of flows, another strategy is needed. There are two generic approaches for modeling dispersed flow: the Lagrangian approach and the Eulerian approach.

For particulate (or particle-like) flow, see Figure 1.1, it is possible to construct methods based on an idea similar to the one behind the MAC-method for separated flow. The general idea is to follow each particle of the flow as they advect in the continuous phase. Here the particles would represent one phase, but not a region as in the MAC-method. This approach is referred to as the Lagrangian–Eulerian method, where the continuous phase is calculated in an Eulerian reference frame. Different strategies exist for implementing this idea; see for example, [1], [5] and [25]. Among these there are three different strategies for coupling between the phases; see [4]. In one-way coupling, the only influence is on the particle by the surrounding fluid. Two-way coupling, the fluid is also influenced by the particle, and in four-way coupling, particles are also influenced by each other.

A different way of modeling dispersed flows is to treat both phases as a continuum. This is generally referred to as the Eulerian–Eulerian approach or the two-fluid model, first discussed by Ishii [16]. In this case local instantaneous equations of mass, momentum and energy balance for both phases are derived along with instantaneous jump conditions for interaction between phases. These equations must then be averaged in a suitable way. A volume concentration or volume fraction function is defined. Using this approach introduces more unknowns than equations, therefore it is necessary to use closure laws, which are often based on empirical assumptions. This model is the one we study, see [12] and [13]. A more detailed description of this method is in Section 1.2.

Kinetic theory describing granular matters or granular flow has also been used to model dispersed two-phase flow. Kinetic theory originates from statistical description of ideal and semi-ideal gases, see [3]. Using this theory it is possible to describe the behavior of molecules or particles with well-defined properties and well-defined interactions. This method introduces a granular temperature, which represents the fluctuation or turbulence of the particle velocity. For further informations see for example [20].
1.2 Two-fluid model - Eulerian–Eulerian

There are several ways, depending on averaging procedure used, to formulate a two-fluid model. A short description follows on how to do this. This description is an overview and based on [2], [7] and [16].

The two-fluid models obtained are generally not of standard mathematical type, at least not for the usual way of closing the system of equations. Results concerning the two-fluid model studied here are discussed in Section 1.2.2.

1.2.1 Derivation of the two-fluid model

The general idea to formulate a two-fluid model is to first formulate integral balances for mass, momentum and energy for a fixed control volume containing both phases. These balances must be satisfied at any time and at any point in space, and gives us two types of local equations. The first type are a local instantaneous equations for each phase and second type expressions for local instantaneous jump conditions, at the boundary between the phases. These set of equations could in principle be solved with numerical simulations, that is if the mesh is finer then the smallest length scales and the time step shorter then the time scales of the fastest fluctuations. However, this is not realistic.

Now for example in particle flow, where there are few particles, it would be possible to track the particles in a Lagrangian way and the carrier or continuous phase in an Eulerian way. This is the Lagrangian–Eulerian approach mentioned in Section 1.1.2. However, this is not always practical, for large number of particles the only practical way is to follow the particle phase in an Eulerian way. Using the Eulerian–Eulerian approach the local instantaneous equations must be averaged in a suitable way, either in space, in time or as an ensemble. These equations can therefore be solved using a coarser mesh and longer time steps, but instead introduces more unknown than the number of equations, therefore closure laws are needed.

The averaged equations

The three most common averaged procedures, that have been used in connection to two-fluid models, are volume, space and ensemble averaging. The ensemble is the most general of these three procedures and the other two can be viewed as approximations of the ensemble averaging. As the names suggest, in the volume average procedure an averaging is done around a fixed point in space at a certain time and similar procedure is for the time averaging. The ensemble average is a little harder to explain, but it can be viewed as the statistical average.

These averaging procedure have their limitations. For example in the case of the volume average, there is a certain restriction on the spatial size where the variables are averaged over. In for example particle gas flow, the characteristic dimension of
the averaging volume must larger than the characteristic dimension of the particle size, but also smaller the characteristic dimension of the physical systems.

Following these procedures the following averaged equations are obtained for a particle flow (or particle-like flow) in a continuous phase under isothermal conditions. Newtonian properties are assumed for both phases. Also, assumptions concerning constitutive and transfer laws for this type of two-phase flow are used. The system is written in the following way

\[
\begin{align*}
(\phi^p \rho_p)_t &= -\nabla \cdot (\phi^p \rho_p \mathbf{u}^p), \\
(\phi^c \rho_c)_t &= -\nabla \cdot (\phi^c \rho_c \mathbf{u}^c), \\
\phi^p \rho_p (\mathbf{u}^p + \mathbf{u}^p \cdot \nabla \mathbf{u}^p) &= -\phi^p \nabla p + \nabla \cdot \mathbf{\sigma} - \mathbf{M} + \phi^p \rho_p \mathbf{g}, \\
\phi^c \rho_c (\mathbf{u}^c + \mathbf{u}^c \cdot \nabla \mathbf{u}^c) &= -\phi^c \nabla p + \nabla \cdot (\mu_c \phi^c \dot{\gamma}^c) + \mathbf{M} + \phi^c \rho_c \mathbf{g}, \\
\phi^p + \phi^c &= 1.
\end{align*}
\]

The system is often closed by

\[
\begin{align*}
\mathbf{M} &= K(\mathbf{u}^p - \mathbf{u}^c), \\
\mathbf{\sigma} &= -\phi^p \rho_p \mathbf{I} + \mu_p \phi^p \dot{\gamma}^p,
\end{align*}
\]

where

\[
\dot{\gamma}^j = \frac{1}{2} \left( \nabla \mathbf{u}^j + (\nabla \mathbf{u}^j)^T \right), \quad j = c, p.
\]

Here \(\phi^p\) is the volume fraction of a particle (or particle-like) phase, \(\phi^c\) is the volume fraction of a continuous phase, \(\mathbf{u}^p, \mathbf{u}^c\) the particle and continuous phase velocity vectors respectively and \(p\) is the continuous phase pressure. Furthermore \(\rho_p, \rho_c\) are the particle and continuous phase densities, \(\mu_p, \mu_c\) are the particle and continuous phase viscosities, \(K = K(\phi^p)\) is the drag function between the phases and \(\mathbf{g}\) the gravity acting on the phases. Finally, \(p_p\) is the particle-collision pressure. Closure laws are needed for \(p_p\) and \(K\).

This model is generally referred to as Model A, there exist other models, for example Model B. The main difference of Model A and B is that in Model B the continuous phase pressure \(p\) is only present in the continuous phase momentum equation (1.4) and not in the particle phase momentum equation (1.3), see for example [10], [21] and [26]. Model A is most commonly used, because Model B is not believed to describe the physics correct.

**Closure laws**

To close the system (1.1)-(1.8), then closure laws are needed for the particle-collision pressure \(p_p\) and the drag function \(K\). Here we will only mention the types of closure laws that are used in this study [12] and [13]. These closure laws are applicable in particle-gas flow, even though general laws of this type are valid in other dispersed flow situations.
In the literature it is common practice to model the gradient of the particle-collision pressure as a function of the volume fraction and it is often written

\[ \nabla (\phi^p p_p) = G(\phi^p) \nabla \phi^p, \]  
(1.9)

where \( G \) can be thought of as the modulus of elasticity for the particle phase. This function is often modeled empirically, there is a huge difference in magnitude between suggested models in the literature, see for example [23] for comparison. In principle, this function vanishes in the dilute limit of small \( \phi^p \) and is practically singular for an upper limit of \( \phi^p \), usually called the maximum packing ratio \( \phi_{mp} \). Due to this last fact, \( G \) is often denoted the hinder function.

In particle-gas flow, the drag force function, that is the momentum transfer between the phases, is often modeled with the following function

\[ K = \left( \frac{17.3}{Re} + 0.336 \right) \frac{\rho_c |u^r|}{d_p} \frac{\phi^p}{(1 - \phi^p)^{1.8}}, \]  
(1.10)

where \( u^r = u^p - u^c \) is the relative velocity between phases and the particle Reynolds number, \( Re \), is defined as

\[ Re = \frac{\rho_c (1 - \phi^p)|u^r| d_p}{\mu_c} \]  
(1.11)

here \( d_p \) is the diameter of the particles. This model is based on the Ergun equation, see [8] and [9].

### 1.2.2 Mathematical characteristics of Model A

Numerous papers have shown that Model A for the inviscid two-fluid formulation in one space dimension (1.1)-(1.8) is only conditionally well-posed, see for example [6], [10], [21], [22], [28], [30]. That is, depending on the data where the 1D two-fluid model is linearized around a well-posed or an ill-posed problem is obtained.

There is a physically fully reasonable region in phase-space were the inviscid case is ill-posed. The periodic inviscid problem is well-posed if the initial data is smooth and fulfills the hyperbolic condition,

\[ G(\phi^p) - (u^r)^2 \frac{\rho_c \rho_p \phi^p}{\rho_c \phi^p + \rho_p (1 - \phi^p)} = \kappa > 0, \]  
(1.12)

and it is ill-posed if \( \kappa < 0 \). In Figure 1.2 the boundary between these regions is plotted, that is \( \kappa = 0 \). Note that in the case of no particle-collision pressure, that is \( G(\phi^p) = 0 \), the inviscid problem is ill-posed were there is a non-zero relative velocity, \( u^r \neq 0 \) and both phases are present, \( 0 < \phi^p < 1 \).

It should be noted that the inviscid Model B is unconditionally well-posed [21], but as mentioned above this model is generally not used.

Recently, it has been shown that the periodic viscous two-fluid formulation is of parabolic-hyperbolic type and therefore locally in time well-posed if the initial
data is smooth, see [30]. The viscous formulation possesses though a medium to high wave number instability.

This means that a smooth solution to the viscous formulation is exponentially unstable, the exponentially growth rate of these instabilities are to the first order

\[
\alpha = \frac{(u_r)^2 \phi^p (1 - \phi^p) \rho_c \mu_c^2 (1 - \phi^p) + \rho_p \mu_p^2 \phi^p}{\mu_p (1 - \phi^p)} - (1 - \phi^p) G
\]

for \( \alpha > 0 \). In practice there exists growth where the inviscid case is ill-posed, \( \kappa < 0 \).

Note here that the magnitude of the growth rate is inversely proportional to the viscosities and it increases quadratically with the relative velocity. In Figure 1.3 the growth rate (1.13) is plotted for particle-gas flow. To comparison, the growth rate for particle-liquid flow, where the density and viscosity ratios are significantly different, would be significantly lower.

Computed solutions to the two-fluid model in one, two and three space dimensions do not explode. How can this be possible?

The first thing one must keep in mind interpreting this result is that it is based on linearization around an assumed smooth solution, followed by localization (freezing of coefficients). The result carries over to the non-linear problem as long as the solution is smooth enough. Loosely speaking, a smooth solution in regions of phase space where \( \alpha \) is moderate or large is exponentially unstable. It is natural to ask how non-smooth the solution can be so that the principle of linearization followed by localization is still applicable?
1.2. Two-fluid model - Eulerian–Eulerian

![Contour plot of the growth rate of the linear instability for particle-gas flow](image)

**Figure 1.3.** Contour plot of the growth rate of the linear instability for particle-gas flow, taken from [13].

Until recently, very little has been investigated about these type of problems. In [18] quasi-linear parabolic problems which are ill-posed in the zero dissipation limit are studied. A simple model problem of this type is considered there

\[
\begin{align*}
  u_t + i \sin(nx)u_x &= \nu u_{xx}, \quad -\infty < x < \infty, \quad t \geq t_0, \\
  u(x, t_0) &= f(x)
\end{align*}
\]

(1.14)

(1.15)

where \(n\) is a natural number and \(\nu > 0\). Freezing of coefficients, \(x = x_0\), gives a growth rate of exponential instability

\[
\alpha = \frac{(\sin(nx_0))^2}{4\nu}.
\]

However, there is a constant \(c\), such if \(|n| \geq \frac{c}{\nu}\), then the solution is uniformly bounded. So, the principle is not valid in this case, see [18] for details.

The are non-linear examples of this type. That is, starting with smooth data, in region of exponential instability, the solution form highly oscillatory structures. These solutions are bounded, see for example [11], [17], [18].

So, there are at least two mechanism that hinder the solution to the two-fluid model (1.1)-(1.8) from exploding like predicted by the analysis of the viscous formulation.

- The solution avoids the region where \(\alpha\) is moderated to large.
- The solution becomes highly oscillatory, or strongly dissipative shock-like structures form.
Chapter 2

Overview of papers

In paper [30] a model problem is studied, that model has the same mathematical properties as the viscous 1D two-fluid formulation. It is illustrated that weak solutions form in finite time. By regularizing the model problem by adding a second order artificial dissipation to the continuity equation gives a computable solution. It was further observed that the solution depends heavily on the amount of artificial dissipation added and no $L_2$ convergence seems to hold for long time simulations, as the artificial dissipation is diminished.

Question is; how similar is this model to the full viscous 1D two-fluid formulation, Model A? Will the full problem behave in a similar way? In paper I [13] we make numerical experiments to investigate this question.

The linear analysis of the 1D viscous problem predicts exponential growth rate of perturbations of smooth data. Computed solutions in 2D and 3D show a highly oscillatory or bubbly behavior, almost of chaotic nature. One may ask; if this behavior is connected by the instabilities seen in the 1D problem? We think that these things are connected. First step toward understanding the instabilities in 2D and 3D is to study the well-posedness of the inviscid 2D model. To our knowledge has this not been done in 2D for this type of models, which is the reason for the analysis in paper II [12].

2.1 Paper I: Numerical experiments with two-fluid equations for particle-gas flow

Here we performed numerical experiments with the viscous incompressible two-fluid formulation, Model A, using closure laws applicable to particle-gas flow. We chose the parameters in these experiments close to physical setup of fluidized bed used in the pharmaceutical industry for coating pellets. Solutions that stay in regions with no or little growth rate are of no interest, there the problem is of standard type.
Different type of initial data were considered, starting in regions where exponential
growth is expected.

In our simulations we use simple non-dissipative numerical schemes, pseudo-
spectral method in space and second order Adam-Bashforth predictor and Adam-
Moulton corrector in time. It was observed that weak solution formed in finite time.
We tried therefore to regularize the problem in a standard way, by adding explicitly
a second order artificial dissipation to the continuity equation of the particle phase.

### 2.2 Paper II: On the well-posedness of the two-
fluid model for dispersed two-phase flow in 2D

Here we investigate the well-posedness of the two-fluid formulation, Model A, for
incompressible inviscid dispersed two-phase flow in two space dimensions. This
problem gives a first order closed system of partial differential equations, with one
of the equation time independent. These equation are therefore not of standard
type. After linearization around constant states and Fourier transformation, one
can however eliminate this equation and obtain a closed system of ordinary differ-
cential equations for each wave number vector.
Couple of interesting things were observed in the numerical experiments performed in paper I [13]. First, that the lower order terms came into play. A strong drag force damps the relative velocity effectively under conditions, when the solution is smooth. This rapid damping gives the balance

$$u^r = \frac{\phi^p (1 - \phi^p) (\rho_p - \rho_c) g}{K(\phi^p)} \quad \text{for } u^r \neq 0$$  \hspace{1cm} (3.1)$$

to high accuracy. However, smooth solutions are not stable, due to the exponential instability. In Figure 3.1 the curve in phase-space defined by (3.1) is plotted, along with the growth rate of the instabilities, this plot is taken from [13]. From simulations we observed that this balance was fulfilled almost everywhere, except in intervals where the solution jumped.

The second thing, was that the linear instability generated highly oscillatory structures or patterns, both in time and space. These patterns were not small and after some time all over the domain. The number of structures seemed to saturate after some time. Furthermore, these patterns where strongly dependent on the regularization parameter, mentioned in Section 2.1. No point-wise convergence was obtain as the regularization parameter was diminished. These results are in agreement with results obtain for a model problem studied in [30].

Now the analysis in paper II [12] gave a similar condition for the well-posedness of the inviscid model in 2D compared to the inviscid 1D result (1.12). If the inviscid 2D model is linearized around constant states $\Phi, U^p, V^p, U^c, V^c$ and $P$, where $\Phi$ is the constant state for the volume fraction of the particle phase, $\phi^p$. Then the 2D model is ill-posed if

$$G(\Phi) < \|U^r\|^2 2 \frac{\rho_c \rho_p \Phi}{\rho_c \Phi + \rho_p (1 - \Phi)},$$  \hspace{1cm} (3.2)$$
where $\mathbf{U}^r = [U^r \ V^r]^T$ is the relative velocity vector. This is consistent with the 1D result, which is obtained by substituting \[ \| \mathbf{U}^r \|_2^2 \rightarrow |u^r|^2. \]
Chapter 4

Future work

There are couple issues concerning the two-fluid model that would be of great interest for further studies in the near future.

It seems impossible to obtain point-wise convergence in 1D, due to the fact that the viscous problem possesses a high wave number instability. The regularization, that is adding second order artificial dissipation, damps the instabilities for the highest wave numbers. Diminishing the artificial dissipation, higher and higher wave number come into play and the solution becomes just more and more oscillatory. However, the solution seems to stabilize in some sense and stay bounded, even if the number of jumps increases.

One may ask if it is possible to obtain convergence is some other sense, if there is some other quantities or measures where convergence of the regularized problem is obtained. A preliminary result concerning the integral of the particle velocity gives some hope, this quantity seems fairly stable. Testing these ideas should not be that difficult for the 1D model.

An analysis of the inviscid 2D model where performed. It is natural to ask if the viscous 2D problem is locally well-posed. And compare with 1D [30].

Such an analysis will only be of local type. To study the long time behavior of the 2D model, numerical experiments will be needed, similar to the work done for the viscous 1D model in [13].
Bibliography


