Parallel Computing of Fluid Flow Through Porous Media

J. Gunnar I. Hellström

Luleå University of Technology
Department of Applied Physics and Mechanical Engineering
Division of Fluid Mechanics
Parallel Computing of Fluid Flow
Through Porous Media

by

J. Gunnar I. Hellström

Division of Fluid Mechanics
Department of Applied Physics and Mechanical Engineering
Luleå University of Technology
SE-971 87 Luleå
Sweden

Luleå, February 2007
Preface

This work has been carried out at the Division of Fluid Mechanics, Department of Applied Physics and Mechanical Engineering, Luleå University of Technology, Sweden during the years 2005-2007.

The research presented here was carried out as a part of "Swedish Hydropower Centre - SVC". SVC has been established by the Swedish Energy Agency, Elforsk and Svenska Kraftnät together with Luleå University of Technology, The Royal Institute of Technology, Chalmers University of Technology and Uppsala University.

Alstom Hydro Sweden, CarlBro, E.ON Vattenkraft Sverige, Fortum Generation, GE Energy (Sweden), Jämtkraft, Jönköping Energi, Mälarenergi, Skellefteå Kraft, Sollefteåforsens, Statoil Lubricants, Sweco VBB, Sweco Energuide, SweMin, Tekniska Verken i Linköping, Vattenfall Research and Development and Vattenfall Vattenkraft, Waprans, VG Power and Öresundskraft are also participating in SVC.

Regarding paper A the authors gratefully acknowledge the support provided by Vattenfall Utveckling AB, the Swedish Agency for Innovation Systems (VINNOVA) and the participating industries in the Polhem Laboratory.

First of all I would like to thank my supervisor Prof. Staffan Lundström for his guidance and support in this work and also my assistant supervisor Dr. Hans Mattsson for his valuable comments.

Secondly, I would like to express my deepest appreciation to the colleagues at the Division, both former and present, for the many fruitful discussions and the pleasant and creative working environment. Especially thanks to Elianne Wassvik, my office companion, for the many giving discussions and help in keeping the office organized.

Finally I would like to thank all my friends and my family for always supporting me during this work.

Summary of contents

Fluid flow through porous media takes place in a variety of technical areas including ground water flow, flow through embankment dams, paper making, composites manufacturing, filtering, drying and sintering of iron ore pellets. When modelling these kinds of flows it is common practice to use averaging techniques rather than computing the detailed flow field in every single pore. This approach is very efficient when averaged quantities are sought for but it is not very convenient for local problems such as forces on particles within an embankment dam and drying of individual iron ore pellets. In this thesis the focus is set on the former and how such forces alter as a function of Reynolds number. Of particular interest are the effects of inertia and turbulence and when they need to be considered. For problem like this a Computational Fluid Dynamics (CFD) approach is well suited since rather complex geometries and flow conditions can be studied by using parallel computing techniques. By using this technique it is also possible to determine at which flow regimes Darcy law is applicable and thus when more elaborate descriptions of the flow like the Forchheimer equation and the Navier-Stokes equations need to be applied. In order to isolate the question above and be able to use advanced models for turbulence a neat geometry is applied being an array of quadratic packed cylinders. To start with the parallel computing capacity of the in-house cluster is scrutinized showing very promising results with up to almost full scalability. Following this study focus is set on the porous media and when inertia-effects need to be taken into account. The significance of this phenomenon turns out to be when Reynolds number is above 10. Then a study in when the flow has to be solved with a full turbulent description has been carried out with the results that as the Reynolds number increases above 100 the significance is clear. In addition a manuscript for a state of the art literature survey is appended.
Appended papers


Additional appended paper of relevance

Paper abstracts

**Paper A:** The design of hydraulic turbine draft tubes has traditionally been based on simplified analytic methods, model and full-scale experiments. In the latest decades, however, the usage of numerical methods such as CFD has increased in the design process due to the rapid escalation in computer performance. Today with parallel computer architectures new aspects of the flow can be considered, but still several problems have to be solved before CFD can routinely be applied in product development. The present paper aims to investigate the parallel performance of a commercial CFD software on a homogeneous networks of computers, being very common solution strategies encountered in the industry today. In addition, the efficiency improvements obtained in earlier experiments by modifying the shape of the draft tube will be considered to deduce if the improvements can be captured with aid of CFD. Result from both the steady and unsteady CFD simulations shows that almost full scalability is obtained with the commercial CFD software CFX-5.7.1. Furthermore, no remarkable improvement in the pressure recovery factor or the flow field is noticed in the CFD simulations as compared to experiments. The discrepancy may be to the applied inlet boundary conditions and/or turbulence model.

**Paper B:** In modelling of flow through porous media inertia-effects must sometimes be considered. This is often done by usage of the empirically derived Ergun equation that can describe the response of several porous media but does not reveal the real mechanisms for the flow. In order to increase the understanding of such flows we have therefore performed a micromechanically based study of moderate $Re'$ flow between parallel cylinders using a Computational Fluid Dynamics approach. The simulations are carried out with quality and trust by using grid refinement techniques and securing that the iteration error is sufficiently small. Main results are that the Ergun equation fits well to simulated data up to $Re' \approx 20$, that inertia-effects must be taken into account when $Re'$ exceeds 10 and that results from stationary simulations replicate time resolved ones at least up to $Re' \approx 880$.

**Paper C:** When modelling flow through porous media it is necessary to know when to take inertia-effects into account as well as when to switch to
a turbulent description of the flow. This is important in a large number of industrial processes such as flow through embankment dams, filtering, composites manufacturing, paper-making and in the refinement of iron ore. From an engineering point of view the problem is often solved with the empirically derived Ergun equation. The drawback with this approach is, however, that the mechanisms for the transitions between the three states of flow are not revealed and time-consuming experiments have to be performed. In order to increase the knowledge of the detailed flow a micromechanical analysis is here carried out by means of numerical studies of flow through arrays of quadratically packed cylinders at a variety of Reynolds number. Main results are that turbulent effects must be taken into account when Reynolds number is larger than 100 while inertia-effects must be considered when the Reynolds number is above 10.
Division of work

The division of the work between the authors for the three papers is presented below.

Paper A

"Parallel CFD simulations of an original and redesigned hydraulic draft tube"
J. G. I. Hellström, B. D. Marjavaara and T. S. Lundström

All the simulations and analysis was performed by Hellström under the supervision by Marjavaara. The paper was written by Hellström, Marjavaara and Lundström.

Paper B

"Flow through Porous Media at Moderate Reynolds Number"
J. G. I. Hellström and T. S. Lundström

The simulations and analysis was performed by Hellström. The paper was written by Hellström and Lundström.

Paper C

"Numerical Study of Turbulent Flow Through Porous Media"
J. G. I. Hellström, P. J. P. Jonsson and T. S. Lundström

The simulations and analysis was performed by Hellström and Jonsson. The paper was written by Hellström, Jonsson and Lundström.
Paper A

PARALLEL CFD SIMULATIONS OF AN ORIGINAL AND REDESIGNED HYDRAULIC DRAFT TUBE
Parallel CFD simulations of an original and redesigned hydraulic turbine draft tube

J.G.I. Hellström 1, B.D. Marjavaara *, T.S. Lundström 2
Division of Fluid Mechanics, Luleå University of Technology, SE-97187 Luleå, Sweden

Received 12 September 2005; accepted 3 August 2006

Abstract
The design of hydraulic turbine draft tubes has traditionally been based on simplified analytic methods, model and full-scale experiments. In the recent years the use of numerical methods such as computational fluid dynamics (CFD) has increased in the design process, due to the rapid escalation in computer performance. Today with parallel computer architectures, new aspects of flow can be considered. However, several problems have still to be solved before CFD can routinely be applied in product development. This paper aims to investigate the parallel performance of commercial CFD software on homogeneous computer networks, as common solutions encountered in the industry today. In addition, the efficiency improvements obtained in earlier experiments by modifying the shape of the draft tube will be considered, to deduce if the improvements can be captured with aid of CFD. Results from both the steady and the unsteady CFD simulations show that almost full scalability is obtained with the commercial CFD software CFX-5.7.1. Furthermore, no noticeable improvement in the pressure recovery factor or the flow field is noticed in the CFD simulations as compared to experiments. The discrepancy may be to the applied inlet boundary conditions and/or the turbulence model.

Keywords: CFD; Parallel computing; Draft tube; Hydropower

1. Introduction
A hydropower plant converts the potential energy of water stored in a dam into electricity by the hydraulic turbine and the generator. In general, one of three types of turbines is chosen depending on the head and discharge of the hydropower plant: either a Pelton, a Francis or a Kaplan turbine. A Pelton impulse turbine is suited for high heads and low discharges, while Francis and Kaplan reaction turbines are more appropriate for medium and low heads, respectively, and high discharges. Turbines are generally highly effective, transforming up to about 95% of the available head into electric energy [1]. Even small efficiency improvements in the turbine are of interest for the industry due to huge production volumes, especially in the runner and draft tube since they are responsible for a large amount of the losses in medium and low headed turbines [2]. The design of these two components has traditionally been based on simplified analytical methods, model tests and full-scale tests. The use of numerical methods such as computational fluid dynamics (CFD) in the design process has, however, increased considerably due to the rapid development in computer technology, and it is an area that will definitely continue to evolve. So far, most of the work has been focused on the runner, but with enhanced computer performance more attention is shifted towards the draft tube and, in particular, to the reliability of CFD simulations in this context [3–8]. This is due to the fact that CFD predictions of the flow field in draft tubes are very challenging and time consuming, caused by complex flow features such as turbulence, unsteadiness, swirl, separation
and secondary flow. Nowadays, with parallel computer architectures, computer power is approaching sufficient and, consequently, CFD predictions of the main flow features are becoming quite accurate. However, models for turbulence still need to be tested and sensitivity to the applied boundary conditions needs to be investigated further. Automatic shape optimisation of draft tubes with the aid of CFD as well as computer optimisation have also been performed recently, where the CPU requirements are even more demanding due to the fact that several CFD simulations have to be evaluated for different draft tube geometries before an optimal design can be found [9,10]. Results from these studies show, however, the potential of using CFD and computer optimisations in the design/rehabilitation process of hydraulic turbine draft tubes. However, several problems also have to be solved here, before the technique can be routinely applied in product development. For example, the stability of the estimated optimum design is often unknown and it is not obvious how to choose the best optimisation technique.

To solve these problems in an efficient and accurate way there exists today an array of CFD software and a variety of parallel computer architectures, using either shared or distributed memory configuration with a homogeneous or heterogeneous network of computers. This paper aims to investigate the parallel performance of a common solution strategy (commercial CFD software + parallel computer architecture) often applied in the industry in order to solve 3D complex flow in a hydraulic turbine draft tube. The chosen CFD software, which is representative of modern numerics, is CFX-5.7.1 from ANSYS [11], and the parallel computer architecture is a homogeneous Windows cluster with dual Intel Xeon 2.4 GHz 32-bit processors, with 2GB ram available on each node and with a 1 Gb/s communication bandwidth network. For decomposition of the grid the MetIS partitioning method is selected, and for communication between the nodes the MPICH message-passing libraries (MPI) are used [11].

From a turbine designer point of view, the surprisingly small efficiency improvements obtained in earlier CFD simulations by Marjavaara [10], compared to experiments by Dahlbäck [12], will be investigated in detail here. In the latter case a total efficiency improvement of about 0.5% was obtained in the turbine, while in the former case an improvement in the pressure recovery of about 0.02% was obtained in the draft tube. The inconsistency can be traced to the assumptions made about the applied inlet boundary conditions, the turbulence model and/or the steady/unsteady flow behaviour [10]. In this paper the steady/unsteady hypothesis will be evaluated by carrying out both steady and unsteady CFD simulations.

The paper is organised as follows: Section 2 describes the two draft tube geometries investigated. Section 3 presents details of the CFD simulations, and in Sections 4 and 5 the results and conclusions of the parallel performance and the CFD analysis are presented.

2. Geometrical description

The main purpose with a draft tube is to convert dynamic pressure into static pressure and thereby increase the efficiency of the turbine. This is done by gradually increasing the cross-section area and consequently decelerating the fluid flow motion, see Fig. 1. A common measure of its performance is the pressure recovery factor $C_p$, being defined as:

$$C_p = \frac{\frac{1}{2} \int_{A_{in}} p \, dA - \frac{1}{2} \int_{A_{out}} p \, dA}{\frac{1}{2} \rho \left(\frac{A_{in}}{A_{out}}\right)^2}.$$  

where $A$ is the area, $p$ is the static pressure, $Q$ is the flow rate, $\rho$ is the density and the subscripts ‘in’ and ‘out’ are correspond to the inlet and outlet, respectively. This measure will be used here in order to compare the draft tube performance with the original and the redesigned draft tube geometry of Hölleforsen Kaplan turbine, located in Indalsälven, Sweden. The original draft tube geometry is well mapped out in two ERCOFTAC Turbine-99 Workshops [4] by using a 1:11 scale model of the real turbine including the spiral case and the draft tube. The design of this draft tube is representative of designs in the 1940s and is characterised by a sharp heel, see Fig. 1. In the redesigned draft tube geometry, suggested by Dahlbäck [12], a removable curved insert has been placed in the sharp heel corner in order to make it smoother, see Fig. 2.

3. CFD simulations

The 3D flow field in the two draft tube geometries are solved with the commercial CFD code CFX-5.7.1, with respectively incompressible, turbulent, steady and unsteady flow assumption. The CFX-5.7.1 solver is based on the finite volume method applied on an unstructured grid. The solver is coupled, meaning that the hydrodynamic equations are solved in a single step [11]. In this case the flow field is calculated based on the Reynolds-Averaged Navier-Stokes (RANS) equations which are derived from the governing Navier–Stokes equations by decomposing the total velocity $\mathbf{u}$ into a mean $\overline{U}$ and fluctuation compo-

![Fig. 1. A CAD model of Hölleforsen draft tube.](image)
The eddy viscosity is modelled by predicting the velocity strain:

\[ S_{ij} = \frac{1}{2} (\partial_i U_j - \partial_j U_i) \]  

The eddy viscosity is modelled by predicting the velocity and length scales \((u \sim l \cdot \ell)\) of the eddy viscosity \((v_F \sim l \cdot \ell)\) with the turbulent kinetic energy \(k\) and the turbulent dissipation rate \(\varepsilon (u \sim k^{1/2} \text{ and } l \sim k^{1/2}/\ell)\)

\[ v_F = C_p \frac{k^2}{\varepsilon} \]  

where \(C_p\) is a model constant. The turbulent kinetic energy and the dissipation rate in Eq. (6) are described by:

\[ \frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i} = -\nabla p + \nu \nabla^2 k - \varepsilon + \frac{\partial}{\partial x_j} \left( \nu + \frac{v_F}{\sigma_k} \frac{\partial k}{\partial x_j} \right) \]  

(7)

\[ \frac{\partial \varepsilon}{\partial t} + U_i \frac{\partial \varepsilon}{\partial x_i} = -C_{1} \frac{\varepsilon}{k} \nabla p \nabla x_j - C_{2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left( \nu + \frac{v_F}{\sigma_f} \frac{\partial \varepsilon}{\partial x_j} \right) \]  

(8)

The parallelisation strategy in CFX-5.7.1 is based on the Single Program Multiple Data (SPMD) model, which basically runs identical versions of the software on several processors. It is designed so that all numerical intensive tasks are run in parallel while the administrative tasks (like
input/output) are performed in a sequential manner by the master process. The communication between the processors is done with either Parallel Virtual Machine (PVM) or the Message Passing Interface (MPI). In this case only the MPICH implementation was tested since it was considerably faster than the PVM implementation. For example, it reduced the execution time per outer solver iteration with about 1.4%, and the preparation and finalisation of the solver solution by more than 100%. A disadvantage of the MPICH as compared to the PVM is that the former is not completely interoperable between all supported platforms.

The decomposition of the grid into a number of partitions, which are then distributed and solved on separate processors, is performed in the CFX-5.7.1 software please see [11]. The selected grids consisted of 1470k and 1445k nodes (respectively, original and redesigned geometry) and had a minimum angle of about 11°, a max. edge length ratio of about 280 and a max. element volume ratio of about 30.

4. Result

4.1. Parallel performance

The evaluation of parallel performance in the CFX-5.7.1 software was performed on a homogeneous Windows cluster with dual Intel Xeon 2.4 GHz 32-bit processors, with 2 GB ram available on each node and with the 1 Gb/s communication bandwidth network mentioned earlier. Only the original draft tube geometry was considered in the evaluation since it is expected that both geometries will give about the same speedup. The parallel efficiency obtained in this case was almost ideal, as shown in Table 1 where the efficiency for the initial, steady and unsteady CFD simulations. An initial guess for the steady calculations was generated by using a first order accurate scheme (First Order upwind) for all equations, and an outlet that allows both in and out flow from the fluid domain. The initial guess for the unsteady calculations was simply the steadiest, one. In addition the outlet of the draft tube geometry was extended downstream, to prevent it crossing a possible recirculation at the actual outlet.

### Table 1

The efficiency (and total execution time) related to the six processor configuration

<table>
<thead>
<tr>
<th># Proc./6</th>
<th>Initial</th>
<th>Steady</th>
<th>Unsteady</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>100 (5.18 × 10^5 s)</td>
<td>100 (6.55 × 10^5 s)</td>
<td>100 (3.80 × 10^5 s)</td>
</tr>
<tr>
<td>1.33</td>
<td>94.75 (4.10 × 10^5 s)</td>
<td>161.07 (3.05 × 10^5 s)</td>
<td>96.61 (2.95 × 10^5 s)</td>
</tr>
<tr>
<td>1.67</td>
<td>98.04 (3.10 × 10^5 s)</td>
<td>173.85 (2.26 × 10^5 s)</td>
<td>101.33 (2.25 × 10^5 s)</td>
</tr>
<tr>
<td>2</td>
<td>95.57 (2.75 × 10^5 s)</td>
<td>167.09 (1.96 × 10^5 s)</td>
<td>97.93 (1.94 × 10^5 s)</td>
</tr>
</tbody>
</table>

Please cite this article in press as: Hellström JGI et al., Parallel CFD simulations of an original and redesigned ..., Adv Eng Softw (2006), doi:10.1016/j.advengsoft.2006.08.013
CFD simulations, indicating that there was a problem with the processor availability, memory availability or communication bandwidth during this simulation [15]. Note that six processors was the minimum number of CPUs necessary to avoid memory swapping in the CFD simulations, and that the same mesh partition was used for the initial, steady and unsteady calculations. The total execution time for the unsteady CFD simulation with 12 processors was about 2 days and 6 h, where the communication time was about 4% of the total time and increased slightly with the number of processors.

4.2. Flow analysis

Results from both the steady and unsteady CFD simulations show small efficiency improvements (pressure recovery) compared to the experiments performed by Dahlbäck [12], but they are in accordance with the CFD simulations performed by Marjavaara [10]. For example, an enhancement in the pressure recovery factor for the redesigned draft tube geometry, compared to the original draft tube geometry, of about 0.006% for both the steady and unsteady CFD simulations, see Table 2. This should be weighed against a total efficiency improvement in the experiments of about 0.5%. It is also noticed that there are small differences between the steady and unsteady CFD simulations, about 0.001%, see Table 2.

Likewise, the velocity and pressure fields are nearly identical in the CFD simulations but in conflict with the experimental result by Dahlbäck [12], see Figs. 3 and 4. The only difference in the flow field is obtained in the sharp heal corner where the separated region disappears in the redesigned geometry, as expected. However, there were no efficiency improvements, see Fig. 5. The CFD simulations performed here also show positive agreement with the simulations carried out in the two ERCOFTAC workshops on the original geometry. Regions with separated flow, secondary flow after the sharp heel corner with two main vortices, and a vortex rope moving from one side of the draft tube to the other is captured, compare [4].

The average $y_+$ values at near wall nodes for the two CFD simulations were about 250, which may reduce the accuracy of the calculations since they are higher than the recommended value of 100 [11]. However most of these values can be traced to the extension of the draft tube, where their effects should not be critical. Removing the extension the average $y_+$ is also getting closer to 100, which

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Steady</th>
<th>Unsteady</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>0.95878</td>
<td>0.95880</td>
</tr>
<tr>
<td>Redesigned</td>
<td>0.95314</td>
<td>0.95319</td>
</tr>
</tbody>
</table>

Table 2
The pressure recovery factor for the original and redesigned geometry

Fig. 3. Streamlines from the runner and velocity contour plots for the original geometry (left) and redesigned geometry (right).

Fig. 4. Pressure distribution at a mid plane for the original geometry (left) and redesigned geometry (right).

Please cite this article in press as: Hellström JGI et al., Parallel CFD simulations of an original and redesigned..., Adv Eng Softw (2006), doi:10.1016/j.advengsoft.2006.08.013
are more satisfactory but not optimal. Each CFD simulation was assumed converged when all root mean square (RMS) residuals had drop to about 10$^{-8}$, which is sufficient for most industrial engineering applications [11].

5. Conclusion

The present paper illustrates the need of parallel computing architectures when performing draft tube CFD simulations due to lack of RAM memory and to reduce the overall computational time. This High Performance Computing requirement is even more essential when the grid becomes finer, when more advanced turbulence models are used and when performing shape optimisations. Using the commercial CFD software CFX-5.7.1, almost ideal speedup can be obtained for draft tube flow simulations, which reduces the execution time satisfactorily. If MPICH communication is also available on the specific platforms used, the execution time can be reduced even further.

The result of the calculated flow field in both draft tube geometries (original and redesigned) agrees well with CFD simulations previously performed by Marjavaara (original and redesigned) [10] and by participants in the two ERCOFTAC Workshops (only original) [4]. Conversely, there were no improvements in the pressure recovery between the original and redesigned draft tube geometry for both the steady and unsteady CFD simulations compared, to the experiment by Dahlbäck [12]. An example of this would be the inlet velocity profile used for the draft tube geometry modifications, which remains fixed during the CFD simulations. This implies that including the runner in the calculations may give better agreement. A finer grid (especially the near wall resolution) and a more advanced turbulence model would certainly also predict the flow field more accurately. Note that the unsteadiness of the unsteady CFD simulation can be discussed in this case, since it is based on the RANS equation, a standard $k$-$\varepsilon$ turbulence model and fixed steady inlet boundary conditions. All of these suggestions imply more expensive calculations, some of which can only be realised by intensifying the parallel computation with the method studied here.

Acknowledgments

The authors gratefully acknowledge the support provided by Vattenfall Utveckling AB, the Swedish Agency for Innovation Systems (VINNOVA) and the participating industries in the Polhem Laboratory that made this work possible.

References


FLOW THROUGH POROUS MEDIA AT MODERATE REYNOLDS NUMBER
Flow through Porous Media at Moderate Reynolds Number

J. G. I. Hellström, T. S. Lundström

Abstract

In modelling of flow through porous media inertia-effects must sometimes be considered. This is often done by usage of the empirically derived Ergun equation that can describe the response of several porous media but does not reveal the real mechanisms for the flow. In order to increase the understanding of such flows we have therefore performed a micromechanically based study of moderate $Re'$ flow between parallel cylinders using a Computational Fluid Dynamics approach. The simulations are carried out with quality and trust by using grid refinement techniques and securing that the iteration error is sufficiently small. Main results are that the Ergun equation fits well to simulated data up to $Re'\leq 20$, that inertia-effects must be taken into account when $Re' > 10$ and that results from stationary simulations replicate time resolved ones at least up to $Re' \leq 880$.

Introduction

Fluid flow through porous media takes place in a number of technical areas including ground water flows, flow through embankment dams, paper making, composites manufacturing, filtering, drying and sintering of iron ore pellets. In several of these areas the flow can simply be described by Darcy’s law, which in its general and one-dimensional form may be written as

$$ u_i = \frac{K_{ij}}{\mu} \Delta p_i, \quad \text{and} \quad \frac{K \Delta p}{\mu} \frac{Q}{L} = \frac{Q}{A}, \quad (1a-b) $$

respectively. In these equations $u$ is the superficial velocity, $K$ the permeability, $\mu$ the viscosity of the fluid, $p$ the pressure, $Q$ the flow rate through an area $A$ and $\Delta p$ the pressure drop over an length $L$ in the stream-wise direction. Darcy’s law is valid as long as $Re$ is sufficiently low, the fluid can be treated as incompressible and Newtonian and the porous medium is fixed. When $Re$ increases over a certain level, which may be the case for erosion in embankment dams and drying of iron ore pellets, the pressure drop becomes higher than what is predicted by (1a-b) and additional non-linear terms are introduced in the so called Forchheimer equation

$$ \frac{K \Delta p}{\mu} \frac{Q}{L} = \frac{Q}{A} + b \left( \frac{Q}{A} \right)^m \quad (2) $$

where $b$ is a property of the porous media and $m$ is a measure of the influence of fluid inertia. The equation was later modified by Ergun by fittings to experimental data according to
\[
\frac{\Delta p}{L} = 150 \frac{(1 - \varepsilon)^2}{\varepsilon^3} \frac{\mu}{A} \frac{Q}{D_p^3} + 175 \frac{(1 - \varepsilon)}{\varepsilon^5} \frac{Q^2}{A^2}
\]

where \( \varepsilon \) is the fractional void volume in the bed and \( D_p \) is the effective diameter of particles [1]. From (3) a modified Reynolds (\( Re' \)) number can be defined

\[
Re' = \frac{\rho D_p}{\mu} \frac{Q}{A} \frac{1}{1 - \varepsilon}
\]

The Ergun equation does not have a micromechanical basis and the geometrical parameters shaping the form of the equation are therefore unknown. Previous works also indicate that the parameters of the Ergun equation may be improved [2, 3].

To increase the knowledge about higher \( Re \) flow through porous media we will here study the detailed flow through an array of quadratic packed parallel cylinders with Computational Fluid Dynamics (CFD). The CFD simulations are performed with the commercial software ANSYS CFX 10.0 with quality and trust regarding grid refinement and iterative errors. To achieve this within reasonable computational time the CFD simulations are run with a parallel computing procedure and by this retaining the iterative error low within reasonable computing time. Another effect that is studied is whether unsteady effects take place at higher \( Re \) and if this effect is of importance for the apparent permeability.

1. Numerical setup

In order to simplify the simulations unit-cells are defined for steady and unsteady flow, see Fig. 1. For both unit-cells, the discretization is performed with ANSYS ICEM CFD 10.0 Hexa where the block-structure is defined and projected onto the geometry to achieve as correct and smooth description of the unit-cell as possible. For the grid refinement study the grid is refined uniformly in the in-plane directions while the number of nodes in the out-of-plane direction is two for all cases. For the unsteady simulations the geometry is mirrored so the whole cylinder is encapsulated in the unit-cell. Consequently the domain is doubled and so is the number of nodes required for a certain mesh density. In order to drive the flow perpendicular to the cylinders a pressure gradient is introduced into the momentum equations by in a sub-domain specifying a general momentum source [4].

Fig. 1. The 2 unit-cells used in the simulations, the left for the steady calculations and the right one for the unsteady.
unit-cells are periodic domain interfaces representing the structure of the array. The advection scheme used to solve the continuity and momentum equations are strictly 2nd order accurate, which is realised by setting the specified blend factor equal to 1 in CFX-Pre. The simulations are considered well converged when the Root Mean Square (RMS) -residuals has dropped 5-6 decades and the max-residuals is no more than 1.5 decade above the RMS-residuals.

For the simulations with Re less than 150 the flow-field is solved by assuming the flow to be steady. At higher Re both steady and unsteady simulations are carried out following the results in the literature [5, 6]. The convergence criteria are the same as for the steady simulations and the mass-flow quantity is stabilized to a constant value.

2. Results

This part is divided into 3 sections dealing with grid refinement, effect on apparent permeability from solid fraction and Re’ and the unsteady calculations evaluated in order to check whether or not it is required to use an unsteady approach in order to get the correct apparent permeability.

2.1. Grid refinement study

The grid refinement study is performed with a solid fraction of 60% and a Re’ of approximately 12 for 10 mesh densities, see Tab 1. The parameter of interest is the apparent permeability which is derived by usage of (1) where \( Q \) is calculated as the mass-flow obtained from CFX-Post divided with the density of the fluid. A polynomial fit to the results indicates that the simulations are in the asymptotic range since the curve approaches a specific value as the grid is refined, see Fig. 2. The simulated values are in all cases close to the asymptotic one with a difference of only 0.3 per mil for the mesh with 370 000 nodes. This accuracy is by all means god enough and since the computational time is short enough for this case all simulations, from now on, are performed with a mesh density in this range.

**Tab.1. Meshes with the corresponding permeability.**

<table>
<thead>
<tr>
<th>Normalized number of nodes</th>
<th>Number of nodes</th>
<th>K</th>
</tr>
</thead>
<tbody>
<tr>
<td>8.72</td>
<td>125 000</td>
<td>7.3839E-08</td>
</tr>
<tr>
<td>5.74</td>
<td>190 000</td>
<td>7.3825E-08</td>
</tr>
<tr>
<td>3.89</td>
<td>280 000</td>
<td>7.3815E-08</td>
</tr>
<tr>
<td>2.95</td>
<td>370 000</td>
<td>7.3810E-08</td>
</tr>
<tr>
<td>2.22</td>
<td>490 000</td>
<td>7.3807E-08</td>
</tr>
<tr>
<td>2.02</td>
<td>540 000</td>
<td>7.3805E-08</td>
</tr>
<tr>
<td>1.73</td>
<td>630 000</td>
<td>7.3803E-08</td>
</tr>
<tr>
<td>1.43</td>
<td>760 000</td>
<td>7.3801E-08</td>
</tr>
<tr>
<td>1.18</td>
<td>920 000</td>
<td>7.3800E-08</td>
</tr>
<tr>
<td>1</td>
<td>1 090 000</td>
<td>7.3798E-08</td>
</tr>
</tbody>
</table>

**Fig. 2. Plot of the different meshes and the corresponding permeability**

\[ y = -1.504E-13x^2 + 6.648E-12x + 7.375E-08 \]

2.2. Different solid fractions

The simulations are performed with solid fractions of 40, 60 and 70% at a number of Re’ ranging from about 0.001 to around 1000. At the creeping flow regime (Re’ < 1) true permeability data are obtained see Fig. 3. As the flow-rate is increased the apparent permeability decreases for all solid fractions but predominantly for the highest one. To shed
some light on this phenomenon the flow field is plotted for two solid fractions and three $Re'$.

Fig. 4 and 5. As seen the circulation zone and the corresponding stagnation points formed move towards the nip as $Re'$ is increased. The circulation zone then becomes small and the velocity near the wall high indicating large losses.

Fig. 3. The result from the simulations with different solid fractions, the apparent permeability is normalized with the true permeability and then plotted against $Re'$.

Fig. 4. Flow fields for a solid fraction of 70% at different $Re'$, 0.001, 80 and 700.
When comparing the simulations at \( f = 60\% \) with the Ergun equation it is obvious that the simulation exhibit fair agreement up to \( \text{Re}' \) of about 20 Fig. 6. where the parameter evaluated are the Blake-type friction factors

\[
f' = \frac{\Delta p}{\rho u^2} \frac{D_e}{L} \left( \frac{\varepsilon}{1 - \varepsilon} \right) \text{ and } f' = 1.75 + \frac{150}{\text{Re}'}.
\]  

(5)

Above this value the slope of the Ergun curve decreases while the gradient of the simulated data stays more or less fixed. This phenomenon indicates that the empirical constants should be upgraded to suit the problem studied better.
2.3. Unsteady simulations

When comparing the steady and the unsteady simulations at \( Re' \) equal to 880 it appears that the final flow fields differs marginally but that the apparent permeabilities are in-principal the same. In addition the residuals for the steady calculations performed do not oscillate, indicating a steady behaviour of the flow. Unsteady calculations when \( Re' \) is less than 880 are therefore not required, which considerably reduces the computational time needed to achieve reliable values of the simulated apparent permeability.

Conclusions

The results from the simulations are verified with a grid refinement study and are also in the same range as results from previous work indicating that the simulations are reliable and can be trusted. This is very important since there are by definition uncertainties with numerical simulations.

The simulations show that the limit of \( Re' \) when inertia-effects must be taken into account when simulating flow through porous media is 10.

The Ergun equation fit well to the simulations up to \( Re' \) about 20, however the discrepancy thereafter shows that the need to update the constants in the Ergun equation is rather obvious.

The simulations show that steady calculations can be used at least up to \( Re' \) equal to 880.

References


Authors
Ph. D. student Hellström, J. Gunnar I.
Prof. Lundström, T. Staffan
Division of Fluid Mechanics
Luleå University of Technology
S - 971 87 Luleå, Sweden
E-mail: gunnar.hellstrom@ltu.se
NUMERICAL STUDY OF TURBULENT FLOW THROUGH POROUS MEDIA
Numerical Study of Turbulent Flow Through Porous Media

J. G. I. Hellström, P. J. P. Jonsson‡ & T. S. Lundström
Division of Fluid Mechanics, Luleå University of Technology, SE-971 87 Luleå, Sweden. Corresponding author: gunnar.hellstrom@ltu.se

Abstract

When modelling flow through porous media it is necessary to know when to take inertia-effects into account as well as when to switch to a turbulent description of the flow. This is important in a large number of industrial processes such as flow through embankment dams, filtering, composites manufacturing, paper-making and in the refinement of iron ore. From an engineering point of view the problem is often solved with the empirically derived Ergun equation. The drawback with this approach is, however, that the mechanisms for the transitions between the three states of flow are not revealed and time-consuming experiments have to be performed. In order to increase the knowledge of the detailed flow a micromechanical analysis is here carried out by means of numerical studies of flow through arrays of quadratically packed cylinders at a variety of Reynolds number. Main results are that turbulent effects must be taken into account when Reynolds number is larger than 100 while inertia-effects must be considered when the Reynolds number is above 10.

Keywords:

Introduction

Flow through porous media takes place in a number of technical areas of application including ground water flow, flow through embankment dams, paper-making, composites manufacturing, filtering, drying and sintering of iron ore pellets. The flow characteristics will however vary as a function of pore geometry and ambient conditions. At one end the fluid is creeping between the pores and Darcy law is fully valid according to:

‡ Currently at Epsilon High Tech AB, Sweden.
\[ u_i = -\frac{K}{\mu} p_{ij} \quad \text{and} \quad \frac{K}{\mu} \frac{\Delta p}{L} = \frac{Q}{A}. \quad (1 \& 2) \]

where (1) represent the general form of Darcy’s law and (2) the one-dimensional one. In these equations \( u \) is the superficial velocity, \( K \) the permeability, \( \mu \) the dynamic viscosity of the fluid, \( p \) the pressure, \( Q \) the flow rate through an area \( A \) and \( \Delta p \) the pressure drop over an length \( L \) in the stream-wise direction. \textit{At the other end} the flow in the pores is fully or partly turbulent and additional non-linear terms are introduced in the so called Forchheimer equation

\[
\frac{K}{\mu} \frac{\Delta p}{L} = \frac{Q}{A} + b \left( \frac{Q}{A} \right)^m \tag{3}
\]

where \( b \) is the property of the porous media and \( m \), in this case, is a measure of the influence of fluid turbulence, Forchheimer (1901). In between the creeping region and the turbulent, laminar inertia gives a substantial contribution to the resistance to flow and experiments have indicated that it is possible to use (3) for this case as well. Since (3) can be applied in a variety of flow situations it is in-place to scrutinize it further.

A modified version of (3) was presented by Ergun by fittings to experimental data according to the following expression

\[
\frac{\Delta p}{L} g = 150 \frac{(1-\epsilon)^2}{\epsilon^3} \frac{\mu Q}{A} + 1.75 \frac{(1-\epsilon)}{\epsilon^3} \frac{\rho (Q/A)^2}{D_p} \tag{4}
\]

where \( g \) is the gravitational constant, \( \epsilon \) the fractional void volume in the bed, \( D_p \) the effective diameter of particles and \( \rho \) the density of the fluid, Ergun (1952). The Ergun equation has shown best agreement with a bed of randomly distributed spheres and is therefore not optimal in for all geometries. The equation has therefore recently been generalised by Nemec & Levec (2005) to yield the following expression

\[
\frac{\Delta p}{L} \frac{1}{\rho g} = \Psi = A \frac{Re^*}{Ga^*} + B \frac{Re^*}{Ga^*} \tag{5}
\]

where \( A \) and \( B \) are tabulated material dependent constants derived from experiments, revealing that the knowledge of the material in itself must be
improved. The Reynolds number and the Galileo number are modified according to Niven (2002),

\[
\text{Re}^* = \frac{\rho \varphi D_p U}{\mu (1 - \varepsilon)} \quad \text{and} \quad \text{Ga}^* = \frac{\rho^2 g \varphi^3 D_p^3 \varepsilon^3}{\mu^2 (1 - \varepsilon)^3}
\]

(6 & 7)

where \( \text{Re}^* \) represent the true averaged velocity in the pores taking the pore volume fraction \( \varepsilon \) into account and including the tortuosity \( \varphi \). The Galileo number in its turn gives us the ratio between gravitational and viscous forces but please also notice the strong dependency on particle diameter and pore volume fraction. Other important contributions for the experimental approach of flow through porous media, Rose (1945), Lesage et al (2004), Wakeland & Keolian, (2003) and Gebart, (1992). From an numerical approach the significant contributions earlier can be considered as the following references, Koch & Ladd (1997), Ghaddar (1995), Edwards et al (1990), Beetstra et al (2007), Papathanasiou et al (2001) and Andrade Jr. et al (1999).

When increasing Reynolds number over the on-set of turbulence the equations to solve become the Reynolds-Averaged Navier-Stokes (RANS) equations. These are derived from the governing Navier-Stokes equations by decomposing the total velocity \( \tilde{u} \) into a mean \( U \) and fluctuation component \( u \) (i.e. \( \tilde{u} = U + u \), where \( U = \overline{\tilde{u}} \)), resulting in the following equations

\[
\partial_i U_j + U_j \partial_i U_i = -\frac{1}{\rho} \partial_i P + \nu \overline{\partial_j \partial_i u_j} \quad \text{and} \quad \partial_i U_j = 0.
\]

(8 & 9)

Here \( \nu \) is the kinematic viscosity and \( \overline{u_i u_j} \) are the Reynolds stresses. These equations represent the mean flow characteristics where turbulent effects are modelled via the Reynolds stresses, in order to obtain closure. When considering flow around a number of periodically arranged cylinders having curved surfaces, as will be done here, the Shear-Stress-Transport (SST), Menter (1993) can be an appropriate choice. In the SST-model the best elements from the \( k-\varepsilon \) and the \( k-\omega \) model are combined via a blending factor. This factor activates the \( k-\omega \) model in the near-wall region and the \( k-\varepsilon \) model in the bulk, yielding that the transition from the \( k-\omega \) to the \( k-\varepsilon \) formulation takes place in the logarithmic part of the boundary layer. The model is based on the assumption that the principal shear-stress is proportional to the turbulent kinetic energy, which is introduced into the definition of the eddy-viscosity, the so-called Bradshaw’s assumption. The mathematical formulations thus develop into
where \( k \) is the turbulent kinetic energy, \( S_{ij} \) the mean strain tensor, \( \omega \) the turbulent frequency, \( \mu_T \) the turbulent dynamic viscosity, \( \nu_T \) the turbulent kinematic viscosity and \( \beta \) \( \sigma_v \) \( \gamma \) \( \sigma_w \) \( \sigma_{vw} \) \( F_i \) and \( \sigma_{w2} \) are constants. For a more elaborate description of the SST-model see Menter (1993).

It is now of interest to study the validity of the SST-model for a variety of Reynolds number for a typical porous medium. It is also relevant to compare the results of laminar set-ups with turbulent ditto. For the purpose of having trustful and high quality results the geometry should remain as simple as possible but still capture pertinent phenomena.

### Geometry and numerical verification

The geometry chosen for this study is an array of quadratically packed infinite long cylinders and in order to reduce the number of volumes to solve the problem a unit-cell is defined, see figure 1. For the calculations the geometry of interest is divided into finite volumes with aid of ANSYS ICEM CFD 10.0 Hexa. The block structure generated is projected onto the geometry in order to get as correct and smooth description of the unit-cell as possible. The number of blocks used in this procedure are eight and for laminar flow it has previously been shown that the quality of the grid it is by all means good enough yielding an error less than 0.3‰ when using 370 000 nodes, see Hellström & Lundström (2006) for a detailed description. To solve the flow the commercial software ANSYS CFX 10.0 is used. The computational domain is parallelized with the MeTis partitioning method and simulated on homogenous Windows and LINUX clusters. For the Windows part of the simulations the MPICH-1.2.5 message-passing libraries (MPI) are used and for the LINUX part a HP-MPI-2.1 routine is chosen, see Hellström et al (2006). For the turbulent simulations measures are taken to keep the y-plus value low enough. To exemplify when Re’ is equal to 2 000 the maximum y-plus is 1.3 being lower than the value recommended in the CFX-manual, the y-plus value should be lower than 2, and also fulfils the y-plus
requirements from ERCOFTAC Best practice Guidelines (2000), which is below 4 and close to unity.

![Figure 1: Schematic sketch of the computational domain.](image)

The boundary conditions of the unit-cell are defined as follows, the top and the bottom part are symmetry-planes, the cylinder wall is assumed to be smooth with a no-slip condition and the left and right hand sides are periodic domain interfaces representing the repeatable structure of the array. In order to drive the flow a momentum source is defined in a subdomain. The advection scheme used to solve the continuity and momentum equations are strictly second order accurate which is realised by setting the specified blend factor equal to one in CFX. The simulations are furthermore assumed to be well converged when the Root Mean Square (RMS) residuals have dropped 5-6 orders of magnitude and when the maximum residuals are less than 1.5 orders of magnitude above the RMS-residuals.

For the unsteady calculations a second order backward Euler scheme is applied and the time step is selected so that the Courant number is between 0 and 5. To discern the unsteady behaviour of the simulations a number of monitor-points were introduced recording pressure at nine locations and also logging the massflow at the domain interfaces. The turbulent simulations were performed with the following turbulence models $k$-$\varepsilon$, Reynolds stress (SSG), SST and a transition model available in CFX. It turned out that the SST-model is the most suitable since it catches the free stream behaviour and since it has a potential to accurately model the flow close to the cylinder wall. In order to shorten the simulation time most simulations were initially based on former runs, in particular each turbulent simulation was based on its laminar counterpart. The turbulent intensity is set as medium (5%) being the default value in CFX, for the eddy frequency the value is set to 300 Hz, based on post-processing of earlier results although the recommendation through the Strouhal number based on the Karman vortex frequency is 15 Hz. The turbulent kinetic energy were chosen to 0.03 J/kg based on post-processing of earlier results, the length scale is a tenth of the diameter of the cylinder and the viscosity ratio is set to 10. To ensure that the
values of the turbulent parameters are in the right region a perturbation analysis were carried out. These variations only weekly influence the results indicating that the solutions obtained are stable in this context.

At higher Reynolds number the simulations were performed with an unsteady approach since the steady simulations indicated that this was necessarily. Investigations for the turbulent case point out that there is not the same need when doing turbulent simulations. This is based on the monitor points for the solver part of the simulation and that the parameters studied show a stable nature for steady simulations.

**Results and discussions**

For the simulation carried out with the full Navier-Stokes Equations but without adding turbulent equations there is a drop in permeability at \( Re' \approx 10 \), see figure 2 where Reynolds number prime is defined with the following formula

\[
Re' = \frac{\rho D_p \frac{Q}{A}}{\mu} \frac{1}{1 - \varepsilon}.
\]  

(12)

As apparent for simulations with solid volume fraction 60 %, the permeability curve then makes a smooth bend and seems to level out. At the high end of this curve it is likely that turbulent effects need to be modelled. Before doing this let us study the flow field calculated for various set-ups.
As $Re'$ is increased the stagnation point on the left hand side of the cylinder climbs “uphill” towards the top of the cylinder and the corresponding recirculation zone becomes smaller and seemingly more chaotic, see figure 3. Hence at very low $Re'$ the circulation generated can be neglected but as $Re'$ increases it becomes important and as shown in the former figure it gives a noticeable contribution to the resistance to flow. The fact that the circulation zone becomes smaller at even higher $Re'$ may explain the result that the apparent permeability levels out, cf. figure 2 and 3.

*Figure 2: The apparent permeability divided by the true permeability for the three solid fractions studied here, 40, 60 and 70%.*
Before deepening this discussion let us compare the simulated results with experiments. It is then convenient to form the Blake-type friction factor according to:

\[ f' = \frac{\Delta p}{L} \frac{D_p}{\rho \left( \frac{Q}{A} \right)^2 \left( 1 - \varepsilon \right) \varepsilon^3} \quad \text{and} \quad f' = 1.75 + \frac{150}{Re'} \quad (13 \ & 14) \]

where (13) is used for the values from the simulations and (14) is a neat form of the Ergun Equation. The correspondence between the simulated value and the Ergun-equation is good until \( Re' \approx 10 \) but as \( Re' \) increases further the simulated results start to deviate from this equation, see figure 4.
In addition the agreement between the simulations and the modified Ergun-equation is very good up to $Re' \approx 10$, but when $Re'$ increases the curves diverge in the same way as before. Whether this is an effect of turbulence or not will now be discussed.

When comparing the turbulent simulations with the laminar equivalents it is shown that the apparent permeability is similar until a certain $Re'$. Increasing $Re'$ further implies that the turbulent simulations give lower apparent permeability values. This is expected and there is no sign of a harsh transition, as the one in pipe flow, in conformity with experimental work presented in Bear, (1972), see figure 5. The deviation is dependent on the solid fraction and for $f = 60\%$ it starts when $Re' > 300$. 

Figure 4: The Blake-type friction factor calculated for the simulations as well as the Ergun equation and the modification by Nemec & Levec for the different solid fractions.
Figure 5: The apparent permeability divided by the true permeability for the three solid fractions 40, 60 and 70%, both the values for the laminar and also the turbulent case.

By studying the averaged velocity field for the turbulent case it can be seen that the circulation zone has more or less the same position for all Re’ but increases in strength with Re’, see figure 6.

Figure 6: Vectors representing the velocity field for the turbulent configuration with different Re’ for the solid fraction of 60%; top left Re’ = 1.5, top right Re’ = 200, bottom left Re’ = 1 000 and bottom right Re’ 2 000.
The most significant difference regarding the flow field occurs on the right hand side of the cylinder where now a clear separation takes place as the circulation zone increases in size. The point of separation climbs closer to the top of the cylinder and causes a more disordered flow scenario as Re’ increases, see figure 7.

Figure 7: The length of the vectors represent the velocity and the velocity gradient, \( \frac{du}{dy} \), is displayed on the cylinder wall for different Re’-numbers: top left: Re’ = 1.5, top right: Re’ = 65, middle left: Re’ = 120, middle right: Re’ = 210, bottom left: Re’ = 1000 and bottom right: Re’ = 2000, when the velocity gradient turn black the gradient switches sign and separation occurs.

When comparing the simulated values with the Ergun and the modified Ergun equation the discrepancy for Re’ > 10 is now reduced although there still is some translation for the simulated values in both the horizontal and vertical direction.
To ensure that the trend presented in figure 8 is general, simulations were performed for the solid fraction 70% as well. The difference between the simulated values and the Ergun equations are larger, but the characteristic bend for the Blake-type friction factor is evident, see figure 9.
Figure 9: The Blake-type friction factor for the Ergun and the modified Ergun equation as well as the simulated values, for the solid fraction of 70%.

When instead plotting the force per unit area acting on the cylinder it is seen that it increases dramatically with Reynolds number but the onset is very much dependent on the solid fraction. It is also evident that a turbulent flow will generate larger forces on the cylinder than a laminar one, see figures 10 and 11. When comparing figures 10 and 11 it is also clear that the normal force stand for the larger part of the force acting on the cylinder.

Figure 10: The shear force acting on the cylinder for different solid fractions.
Conclusions

The main result of this investigation is that the Shear Stress Turbulence model can predict the main flow features in a good way. Furthermore, inertia-effects must be taken into account when Reynolds number is above 10 and the turbulent effects ought to be considered when Reynolds number is increased above 100. The normal force on a cylinder dominates over the shear forces and it is clear that it is dependent on the solid fraction as well as if the turbulence is modelled or not.

Acknowledgement

The research presented was carried out as a part of "Swedish Hydropower Centre - SVC". SVC has been established by the Swedish Energy Agency, Elforsk and Svenska Kraftnät together with Luleå University of Technology, The Royal Institute of Technology, Chalmers University of Technology and Uppsala University. www.svc.nu

References


CFX® (Copyright© 1996-2005), Version 10.0, ANSYS Europe Ltd.


Report A

Fluid Mechanics of Internal Erosion of Embankment Dams
Introduction

We will here briefly outline the status of fluid mechanics of internal erosion in embankment dams. Of particular interest is models for fluid flow generated forces on particles located in porous beds. This condensed review will be divided into two parts where part one deals with homogenous materials for which averaged expressions such as Darcy’s law and Ergun Equation can be applied. This includes a general background to porous media flow, continuum models for flow through embankment dams and routes to numerically model the flow. In part two, models to derive forces on individual particles are discussed starting from dilute (single particle) systems and ending with very dense and periodic systems of particles. The review is then finalised with a discussion. Let us however start with a presentation of some scenarios for internal erosion in embankment dams and please notice that the review has no intention of being complete. It rather aims at given a flavour of the area hopefully inspiring the reader to search for more information from other sources some of which are presented in the list of references.

Internal erosion processes

Internal erosion in embankment dams is not a completely understood phenomenon. Dam failures, accidents and deterioration of dams caused by ageing do take place as a result of internal erosion. Even if dams generally should be considered as safe constructions, a large part of the present knowledge about problems associated with internal erosion is a result of studies of former dam
incidents. This circumstance is almost inevitable since internal erosion processes are very complex and have effect inside the embankment or foundation making the erosion transparency limited until it has progressed enough to be visible or detected by measurements.

The soil fractions that are considered as most susceptible to erosion are relatively uniform coarse silt and fine sand. Cohesive soils as clays are more resistant to erosion as long as the chemical bonds are not destroyed, Srbulov (1988). It seems like some core materials of glacial origin can be particularly susceptible to internal erosion. Glacial till is of special interest since it is a common core material in Swedish dams. Erosion susceptibility of glacial till is described in e.g. Foster et al. (2000), Ravaska (1997) as well as Norstedt & Nilsson (1997).

According to Fell et al. (2005), four conditions must exist for internal erosion and piping to occur. These are:”1) there must be a seepage flow path and a source of water; 2) there must be erodible material within the flow path and this material must be carried by the seepage flow; 3) there must be an unprotected exit, from which the eroded material may escape and 4) for a pipe to form, the material being piped, or the material directly above, must be able to form and support “roof” for the pipe”. Hence for internal erosion to occur particles within the dam must be subjected to forces high enough to make them move. It is therefore in-place to determine the magnitude and direction of the hydrodynamic forces generated by the flow on different types of particles at a variety of conditions.

In a central core earth and rockfill dam there are mainly three processes, Fell et al. (2005), which can initiate piping: backward erosion, concentrated leak and suffusion. Backward erosion is initiated at the exit point of seepage and the erosion is gradually progressing backward forming a pipe. Concentrated leak initiates a crack or a soft zone emanating from the source of water to an exit point. Erosion gradually continues along the walls of the erosion hole causing the concentrated leak. Suffusion is the process where the fine particles of the soil wash out or erode through the voids formed by the coarser particles. This can be prevented if the soil has a well graded particle size distribution with sufficiently small voids.

Internal erosion and piping might occur in the embankment, in the foundation and from embankment into the foundation, see Fell et al. (2005). Failure statistics presented by Foster et al. (2000) show that piping through the embankment is the most common piping mode of failure of those mentioned above, while piping from embankment into the foundation is the least common. In Figure 1, backward erosion piping through the embankment is sketched.
It is reasonable to assume, that under normal conditions processes of internal erosion and piping may be in progress in dams to a smaller extent without any accidents or failures as result. However, potential breach mechanisms (Fell et al., 2005) for extensive internal erosion and piping might be: "gross enlargement of the pipe hole, unravelling of the toe, crest settlement or sinkhole on the crest leading to overtopping and instability of the downstream slope".

A concentrated leak in the embankment (Fell et al., 2005) may be initiated by e.g. cracking due to differential settlements, hydraulic fracturing, and soft zones with high permeability as well as piping modes associated to the presence of conduits or walls. There are a lot of different scenarios in connection to the development of cracks because of differential settlements, see e.g. Fell et al. (2005) and Kim et al. (2004). Hydraulic fracturing, see e.g. Ng & Small (1999) and Sherard (1986), can occur in a core if the vertical effective stress is reduced enough to allow the water pressure to create a crack or a soft zone. This can for example happen if the core settles more than the material in adjacent zones and thereby “hangs up” on it, with a decrease in the vertical stress locally in the core as a result. Soft zones with high permeability in a dam can have several reasons, e.g. segregation of materials. Segregation is the process that causes separation of a graded soil material into finer and coarser zones. It might occur at placement of material by downslope discharge during dumping and spreading operations. The larger particles have a tendency to accumulate at the bottom of the slope. Milligan (2003) warns that the issue of segregation is a critical uncertainty that is generally poorly recognized and dealt with in dam engineering. Foster et al. (2000) points out that many piping failures and accidents through the embankment are associated with the presence of conduits. In Fell et al. (2005) three different modes of such piping are described: piping into the conduit, piping along and above the conduit as well as flow out of the conduit.

Some common observations during piping incidents in embankment dams are listed in Fell et al. (2005) as: excess pore pressures, whirlpool in reservoir, sand boils, cracking, settlements, sinkholes, muddy leakage and increase in leakage. Further, Fell et al. (2005) states that "it is apparent that many accidents would have become failures if they had not been detected by monitoring and surveillance and some action taken". According to Fell et al. (2005), the most
important detection methods and items to monitor in dam surveillance are: "inspection by trained observers, seepage flow, vertical and horizontal displacement by direct survey of surface points, pore pressures as well as reservoir water level and rainfall (as these link to the other factors)".

In recent years, numerical methods have been used to an increasing degree for solving more complicated problems in geotechnical engineering. A wide range of problems have been analysed with e.g. the finite element method. By utilizing the finite element method, based on a continuum approach, stresses, strains, deformations and pore pressures in an embankment dam can be theoretically analysed and it might be possible to draw conclusions from that information about the initiation of internal erosion processes. An approach worth considering, when initiation is established, is to interpret internal erosion numerically as a type of localisation and describe the constitutive behaviour with micromechanical models in localised zones. The mathematical description of the strain localisation phenomenon is nowadays an intensive research area (see e.g. Adachi et al., 1997) and, hopefully, models for strain localisation could give valuable input to the formulation of internal erosion localisation. Such a numerical software could be useful for: increasing the knowledge about internal erosion processes, evaluating the risk for dam incidents caused by internal erosion, estimating the time for progression of internal erosion and piping, studying self-healing of leaks and ageing effects in dams, analysing the amount of instrumentation needed and the proper location for monitoring and surveillance, designing dams etc.

**Continuum models**

Flow through porous media is generally modelled by a continuum approach implying that the flow in the pores is averaged and that averaged quantities such as the permeability are introduced.

**General background**

The flow through porous materials is often described by Darcy’s law that gives a linear relationship between the flow rate and the pressure. The law was originally derived from experiment (Bear 1972; Scheidegger 1972; Dullien 1992) but Tucker and Dessenberger have shown that the law can be derived from the Navier-Stokes Equations (Tucker III 1994). The conditions are that inertia and long-range viscous effects are negligible of which the former is generally a too
crude assumption for at least the final stages of internal erosion. Let us, however, for the moment stick to the stated assumptions and write Darcy’s law in its general form:

\[ v_i = \frac{K_{ij}}{\mu} p_j, \]  

(1)

where \( v \) is the superficial velocity, \( K \) the permeability tensor, \( \mu \) the viscosity and \( p \) the pressure. An obvious interpretation of (1) is that the permeability can vary as a function of direction. It is also important to notice that:

\[ u_i = \frac{v_i}{\phi}, \]

(2)

where \( u_i \) is the average velocity in the pores and \( \phi \) the porosity. In this rigorous form of Darcy’s law the permeability is solely dependent on the detail geometry of the porous material and has the unit \( m^2 \). In the literature describing embankment dams it is also common to give the permeability the unit \( m/s \), conductivity or velocity, Fell et al (2005). This implies that the viscosity and the gravitational constant are incorporated in the actual permeability value. The advantage of this approach is that a velocity-like component is explicitly given. The draw-back is that the actual value is dependent on the conditions for the measurements.

The permeability of a porous material is generally obtained from measurements although numerous models often based on the microgeometry of the porous media have been derived. The models are developed for special cases and measurements need to be carried out in order to calibrate the models (Gebart 1992). Many methods for the experimental assessment of the permeability tensor
have been proposed since the experiments by Darcy, (Kershaw 1972; Adams 1988; Pikulik 1991; Parnas 1993; Parnas 1995; Young 1995; Gebart 1996; Lekakou 1996; Weitzenböck 1998; Lundström 1999; Lundström 2000). Most of these methods are based on either of two fundamentally different principles: (i) **parallel flow**, by which the liquid is made to flow in a controlled direction through the porous sample (the Darcy experiment, see Figure 2); and (ii) **radial flow**, by which the liquid is injected at a “point” and flows freely in all directions (usually in a plane). Either of these principles may be applied in a variety of ways, e.g. by driving the liquid by a constant flow rate or a constant pressure drop. The advantage of the radial flow techniques is that a full permeability tensor, including principal directions, can be obtained in a single experiment (Ahn 1995; Weitzenböck 1998). The disadvantage is that they are based on tracking a flow front, i.e., the flow must be unsaturated, with the risk of including various transient effects, such as capillary action and void formation at the wetting flow front. Parallel flow measurements, on the other hand, can be carried out under steady state conditions (Lundström 2000) and edge effects can be avoided (Lundström 2002). Hence there are by all means several ways to determine the overall permeability of a porous material.

As Re increases over 1-10 it is well-known that experimental data starts to deviate from Darcy’s law. The overall effect is that the pressure drop becomes higher than what is predicted by (1-2) due to inertia and turbulence and an additional term is introduced in the so called Forchheimer equation

\[-\frac{1}{\mu} \frac{K \Delta p}{L} = \frac{\beta}{A} + b \left( \frac{Q}{A} \right)^m \]  

where \( b \) is a property of the porous media and \( m \) is a measure of the influence of fluid inertia. The equation was later modified by Ergun by fittings to experimental data according to

\[ \frac{\Delta p}{L} g = 150 \left( 1 - \epsilon \right)^2 \frac{\beta}{\epsilon^3} \frac{Q}{A} \frac{\rho}{D_p^2} + 1.75 \left( 1 - \epsilon \right) \frac{\rho Q^2}{\epsilon^3 D_p} \]  

where \( \epsilon \) is the fractional void volume in the bed and \( D_p \) is the effective diameter of particles (Ergun 1952). The Ergun equation does not have any micromechanical basis and the geometrical parameters shaping the form of the equation are therefore unknown. Recent studies indicate that the parameters of the Ergun equation should be modified for certain geometries (Papathanasiou 2001;
Nemec 2005) which was also confirmed by a study by Hellström and Lundström (2006).

**Flow through embankment dams**

The flow through embankment dams can be modelled in numerous ways as outlined in Bear (1972). It can, for instance, be convenient to apply Hodographic methods in which the boundaries of the fluid including the free surface within the dam are transformed to form simpler geometries. Although the flow in itself can be treated with explicit expressions such as Darcy’s law non-homogeneous and anisotropic permeabilities and the naturally formed free surface often makes realistic modelling with analytical tools complex.

One possible way to model the flow through embankment dams is instead to solve for the momentum Equations for two phases and then add an advection Equation to deal with the motion of the surface formed between the phases, the air and the water, for instance. To deal with the sharp transition in viscosity and density a modified Heaviside-step function can be introduced, see (Sharif et al 2001). Adaptive FEM analysis of free surface flow is also applicable, (Chen Sheng-Hong, 1996). Another method is to combine spectral analysis with a Laplace transform in order to model the transport of material through for instance the riprap, (Wörman & Xu, 2001). A combination of the Finite Element Method with a stochastic approach and the FEM part is used to solve the mechanical behaviour of the earth structure while the stochastic method is used for the probabilistic part of the earth modelling. (Mellah et al, 2000). Numerical procedures based solely on the finite element method are also common. Such a technique can, for instance, be applied to for hydraulic fracturing in the core of earth and rock-fill dams. The finite element procedure makes then use of special joint elements that allow fluid flow and fracture to be modelled. This technique allows the progress of pore pressure in the core of a dam to be simulated, (Ng & Small, 1999). The Finite Volume Method can be used to solve the flow through the porous material. With such approaches the main flow features are resolved in a better way (Yang & Hemström, 1999, Eriksson & Yang, 2005).

**Micro mechanical models**

Continuum formulations of flow through porous media often results in Equations consisting of a few unknown parameters such as the permeability. The physical
background of such parameters can often be traced to the detailed flow in the pores and it is therefore in place to study the flow on this level, as well. To start with we can scrutinize models for single particles. The viscous drag force on a single sphere in an infinite fluid at \( \text{Re} < 1 \) is, for instance, described by Stokes law:

\[
f = 6\pi R \mu v
\]  
(5)

where \( R \) is the radius of the sphere and \( v \) the velocity of it. This expression was early extended by Oseen to be valid for \( \text{Re} \) up to about 10 according to Lamb (1932):

\[
f = 6\pi R \mu v \left(1 + \frac{3}{16} \text{Re}\right)
\]  
(6)

and later on by Happel and Brenner (1957) to be valid for the case of a sphere moving through a tube with diameter \( D \) to be:

\[
f = 6\pi R \mu v \left(1 + 2.1 \frac{R}{D}\right)
\]  
(7)

It has also been derived for particles with other geometries than spherical. For slender rods the drag force per unit length along the rod is, Emersleben (1925):

\[
f = 4\pi \mu v
\]  
(8)

While the following expression has been derived for flow perpendicular to the rod, Lamb (1932):

\[
f = \frac{4\pi}{2 - \ln(\text{Re})^2} \mu v
\]  
(9)

The obvious conclusion is that the drag on a particle is linearly dependent on the averaged velocity as long as \( \text{Re} \) is less than unity. The importance of velocity then increases to 2\(^{nd}\) order.

For very dense systems and at low \( \text{Re} \) the flow follows Darcy’s law and the porous media can be characterised by their permeability through measurements or by the usage of permeability models based on, for instance, the detailed rod geometry: cf. (Speilman 1968; Bear 1972; Scheidegger 1972; Sangani 1982; Jackson 1986; Dullien 1992; Wang 1994; Higdon 1996). Generally permeability
models may be divided into the following models, Phenomenological, Conduit flow (Kozeny-Carman for instance), Empirical, Network, Deterministic (directly from Navier-Stokes) and models based on flow around submerged objects (models proposed by Brenner and Brinkman, for instance) (Dullien 1992).

The permeability is often represented by the Kozeny-Carman relation, according to:

\[ K = \frac{1}{B \frac{\varepsilon^3}{S^2(1-\varepsilon)^2}}, \]  

where \( \varepsilon \) is the porosity, \( B \) a constant and \( S \) the specific particle surface. The Kozeny-Carman relation is based on a tube-like representation of a porous medium. The constant \( B \) has to be determined experimentally, but has never been determined explicitly Williams (1974). The Kozeny-Carman relation assumes an isotropic porous medium. For rods other Equations have been derived that distinguish between the permeability along and perpendicular to the direction of the rods, according to:

\[ K_1 = \frac{8}{c} \frac{\varepsilon^3}{(1-\varepsilon)^3} R^2, \]  
\[ K_\perp = C \left( \sqrt{\frac{\varepsilon_{\min}}{\varepsilon}} - 1 \right)^{5/2} R^2 \]

where \( R \) is the radius of the rods and \( c \) and \( C \) are constants depending on the arrangement of the rods while \( \varepsilon_{\min} \) denote the minimum pore volume for this arrangement. Also in this case forces on individual particles can be calculated. Mei & Auriault (1991) derived, for instance, the average drag acting on a particle in a porous medium with fore-aft symmetry at small Reynolds number to be:

\[ f = \mu \nu \left( k_0 + k_2 \text{Re}^2 \right) \]

for a two-dimensional array. Here \( k_0 \) and \( k_2 \) are non-dimensional constants to be determined which is, for instance, done for both ordered and random arrays of spheres by Hill, Koch & Ladd (2001). To summarize the forces on individual particles have been exploited for certain geometries and for a number of flow conditions. We however need to investigate further higher Reynolds number flows, more complex geometries and instationary conditions.
Conclusions

To conclude fluid mechanics of internal erosion in embankment dams is rather complex independent on the scale studied. It is therefore of highest importance to do detailed studies of the flow to clarify the mechanisms behind the starting of the process, the development of it and then conditions for healing or complete progress to breakage. Such studies must then be coupled to formulations on the continuum scale giving macroscopic conditions.

Acknowledgement

The research presented in this thesis was carried out as a part of "Swedish Hydropower Centre - SVC". SVC has been established by the Swedish Energy Agency, Elforsk and Svenska Kraftnät together with Luleå University of Technology, The Royal Institute of Technology, Chalmers University of Technology and Uppsala University.

Alstom Hydro Sweden, CarlBro, E.ON Vattenkraft Sverige, Fortum Generation, GE Energy (Sweden), Jämtkraft, Jönköpings Energi, Mälarenergi, Skellefteå Kraft, Sollefteåförsens, Statoil Lubricants, Sweco VBB, Sweco Energuide, SweMin, Tekniska Verken i Linköping, Vattenfall Research and Development and Vattenfall Vattenkraft, Waplas, VG Power and Öresundskraft are also participating in SVC.

References


