PART 1: FLUID-STRUCTURE INTERACTION Simulation of particle filtration processes in deformable media

Marianne Mataln[1], Gernot Boiger[1], Bernhard Gschaider[1] and Wilhelm Brandstätter[2]
[1]ICE Strömungsforschung GmbH, Hauptplatz 13, 8700 Leoben, Austria
[2]Montanuniversität Leoben, Franz-Josefstrasse 18, 8700 Leoben, Austria
www.ice-sf.at

ABSTRACT
In filtration processes it is necessary to consider both, the interaction of the fluid with the solid parts, as well as the effect of particles carried in the fluid and accumulated on the solid. In this first part a closer look is taken on the influence of the fluid on the solid regions. The required algorithm to couple the governing differential equations is derived on the basis of the equations, governing the solid and fluid regions. For discretization, only one single computational mesh is used and this is adjusted to the deformation at each time-step. The simulation of the fluid-structure interaction is realised in a single finite volume flow solver on the basis of the OpenSource software OpenFoam.

Keywords: Simulation; Filtration processes; Fluid-structure interaction; OpenFoam

1. IN GENERAL

Filtration processes arise in a wide variety of industries. Examples include the automotive industry, sewage filter systems, or even facilities present in our household, e.g. the coffee machine. Filters consists of a maze of different fibres. On the surface of those fibres, dirt particles can settle. It is sensible to improve filters in every possible form to reach higher efficiencies. These improvements can be achieved by experiments or simulations. In the majority of cases, experiments on filters are carried out with destructive tests. In that case filters are cut or burned to gain the searched quantities. Yet these tests cannot be repeated in order to obtain other characteristics of the filter or to prove a different outcome. Another disadvantage of experiments is the difficulty to investigate filter material in detail, i.e. to find out the influence of the deposition of particles or the deformation of fibres on the overall filter characteristics. Therefore simulations can offer an attractive alternative to experiments. Every quantity of a filter can be investigated, displayed and processed for further usage. The virtual tests can be repeated as often as necessary with exactly the same initial situation and properties.

The technical application in the present study refers to oilfilters, shown in Figure (1), being in widespread use within the automotive industry. Their main purpose is to filtrate the engine wear from the motor oil, which is used to lubricate the components of the engine.

Figure 1 Detailed view of an Oilfilter.

In modelling of such a filter material, first it is required to investigate the main governing factors of the filtration process. One of those is, that the flow of the fluid through this filter induces a pressure distribution on the fibres, which leads to a deformation of the solid parts. This deformation results in a different assembly of the fibres, which will have significant impact on the other factors, as well as on the overall material. Another effect results from the fact that the fluid is carrying abrasive particles resulting from engine wear. The efficiency is measured on the ability of the filtermaterial to absorb different classifications of particles. The analysis of the fluid-structure interation is essential since it can change the filter characteristics significantly and therefore has to be investigated.

For simulation of such a filter not all factors, which govern fluid flow, have to be considered. A few simplifications can be made in order to reduce the time and effort required for computational fluid dynamics (CFD) simulations. One example is that only a microscopic piece of the filtermaterial is modelled and then extrapolated to the overall filter performance. Due to the size of the geometry, which is in the scale of a few microns, the Reynolds numbers are very low. In that case the flow can be classified as creeping flow and turbulence effects have not to be taken into account. Temperatur also has little influence on the deformation of the filtermaterial and the existing particles and therefore the model can be assumed to be isothermal. There is only oil to be considered as a single fluid and hence single phase flow assumptions are valid as well. Generally not very high pressures are occuring during the filtration process and thus changes of the oil density are not significant. From this follows, that the flow can be assumed to be incompressible.

Now the main task is to investigate how the characteristics of the oilfilter will change with the deformation of the fibres due to the flow of oil. It is essential to understand the underlying driving factors for further improvement of filter materials.

2. THE PROGRAMMING PLATFORM: OPENFOAM

OpenFOAM [1] is a collection of different solvers and applications, which is applicable for a wide range of problems in continuum mechanics. It is not a programm, but more of a toolbox. The other strength of OpenFOAM is that it is an OpenSource [2] application. This allows modifications of the underlying source code for existing problem cases and even permits the creation of whole new solvers, if necessary. This makes OpenFOAM a very flexible tool, which in addition is free of license costs. The underlying programming language is the object oriented C++ programming language, which allows easy and direct implementation of new software modules. However the user has to have an a priori knowledge of the physics and programming techniques to avoid major possible mistakes.

In OpenFOAM the solver applications utilise a syntax similar to the differential equations being solved. For example the momentum equation for fluid flow can be programmed as such:

Algebraic:

$$\frac{\partial \rho U}{\partial t} + \nabla \cdot \phi U - \nabla \cdot \mu \nabla U = -\nabla p \tag{1}$$

Source Code:

```
solve
(
                        fvm: :ddt(rho, U)
                      + fvm: :div(phi, U)
                      - fvm: :laplacian(mu, U)
                        ==
                      - fvc: :grad(p)
);
```

where by $\phi = \rho \cdot U$.

For the above reasons, OpenFOAM is the best environment to create new solvers and hence model multiphysics problems such as the fluid-structure interaction found in oilfilter applications.

3. FLUID-STRUCTURE INTERACTION

3.1. INTRODUCTION

In modelling of filter media a very important physical effect has to be considered. It concerns the interaction between the fluid and the fibre material. The fluid flow induces forces on the solid regions, which lead to deformation of the fibres and hence to overall material structure changes under working conditions. This induces severe changes in the permeability of filters. It is therefore of upmost importance to understand the underlying physical phenomena.

For modelling fluid flow, CFD codes are commonly employed and for mechanical stresses CASA (computer aided stress analysis) codes are widely used. The CFD codes are based on the so-called Finite Volume Method, whereas the CASA codes are commonly based on Finite Element principles. This normally requires that two different computer codes are needed to calculate both, fluid flow and solid-stress simulation for the same system. It implies that both codes must be available and the user must have knowledge of both of them. Another problem is that as a matter of fact the CFD code must be converted into an appropriate code readable by a CASA programm and vice versa for the case that the stresses influences the flow of the fluids. For both circumstances this conversion will cause a certain degree of loss in accuracy.

3.2. A SINGLE ALGORITHM

The interfacing between the two different codes is rendered unnecessary, when it is possible to develop a single computer code, which can then solve the solid-stress equations and displacements in one part of the field and the ones for fluid-flow, i.e. fluid velocity in another. If a closer look is taken at the governing equations for solid-stress and fluid flow, certain similarities can be detected [3]. It is possible to couple the two equation systems and develop one single computer code, as shown below.

3.2.1. Governing equations for fluid flow

The flow of any fluid is described by the fundamental equation system, namely the NAVIER-STOKES equations, which governs the conservation of mass

$$\frac{\partial \rho}{\partial t}+\frac{\partial\left(\rho u_i\right)}{\partial x_i}=0 \tag{2}$$

and momentum:

$$\rho \frac{du_i}{dt}=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\cdot\left(\mu\cdot\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k}\right)\right)+\rho g_i \tag{3}$$

where by u_i an u_j are the fluid flow in coordinate directions x_i, p represents the pressure on the fluid cell, μ the dynamic viscosity, ρ the fluid density and g_i the gravity. [4]

Furthermore, $\delta_{ij} = 0$ for $i \neq j$ and $\delta_{ij} = 1$ for $i = j$.

For incompressible flows the following is valid: $\frac{\partial u_k}{\partial x_k}=0$

3.2.2. Governing equations for displacement in solids

The mathematical model describing the deformation of solids is based on the three dimensional stress distribution. [5]

$$\frac{\partial}{\partial x_j}\left(\sigma_{ij}\right)=0 \tag{4}$$

where for $i = j$, σ_{ij} represent the normal stresses and for $i \neq j$, σ_{ij} represent the shear stresses T_{ij} respectively.

After rearranging and applying HOOKE's law these equations can be written as

$$0=\frac{\partial}{\partial x_j}\cdot\left(G\cdot\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2\nu}{1-2\nu}\delta_{ij}\frac{\partial u_k}{\partial x_k}\right)\right)+X_i \tag{5}$$

where by u_i an u_j are the fluid flow in coordinate directions x_i, G represents the shear modulus, ν the Poissons ratio and X_b are external forces.

Furthermore, $\delta_{ij} = 0$ for $i \neq j$ and $\delta_{ij} = 1$ for $i = j$.

Again, for incompressible flows the following is valid: $\frac{\partial u_k}{\partial x_k}=0$

3.2.3. Coupling

A lot of similarities can be found, when comparing the governing equations for displacement in solids and the velocities of fluids.

At a first glance these two equation systems almost look identical. The main difference between them is the pressure gradient term of the NAVIER-STOKES equations, Eqn. (3),

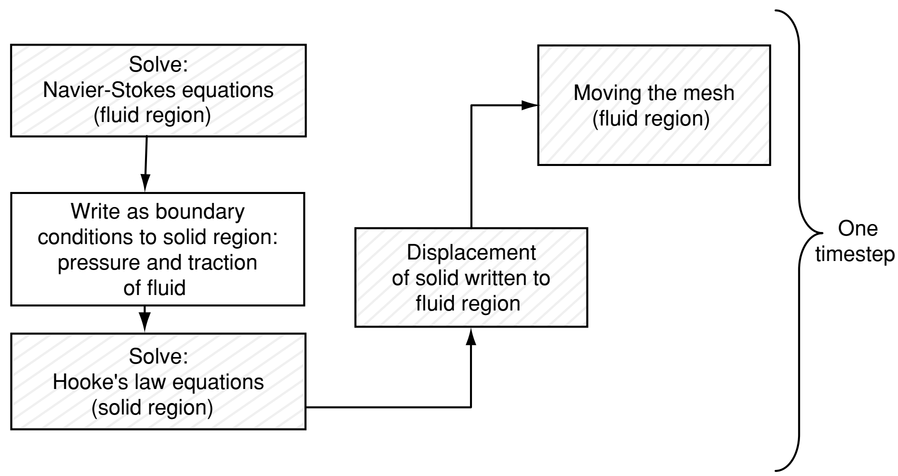

Figure 2 Algorithm to model Fluid/Structure interaction.

which is absent in the displacement equations for solids. This term can be seen as an external force represented by the term X_b in the solid displacement equations. As such coupling between the fluid flow and solid-stress equation systems can be directly achieved. The pressure field of the fluid region can be mapped as a boundary condition on the solid region and hence added as an external force to the term X_b in Eqn. (5). This induces deformations in the solid regions, i.e. the fibre material.

The resulting algorithm is displayed in Figure (2).

3.3. MOVING OF THE COMPUTATIONAL MESH

Essential for successful application of the algorithm, shown in Figure (2), is an efficient method to compensate for the deforming geometry of the solid region, i.e. a moving computational mesh strategy for the fluid region. This movement of the mesh and hence the coordinates of its grid points is based on the Laplace equation:

$$\nabla \cdot \left(D \cdot \nabla \vec{u} \right) = 0 \tag{6}$$

where by u represents the grid velocity and D a diffusion coefficient. The propagation of the deformation values of single cells to the overall cell-collective is carried out by a diffusion mechanism, which is ruled by this diffusion coefficient D. There are different coefficients available in OpenFOAM.

3.4. COMPLETION OF THE MODEL

The fluid forces X_b acting on the fibre material can be decomposed into pressure based (normal) and a shear stress based (tangential) forces, as shown in Figure (3).

In addition to the pressure force F_p defined in Eqn. (7), the flow of the fluid with the free stream velocity u_∞, also induces shear forces F_τ on the solid region. These forces are caused by velocity gradients normal to the solid surface and also have a major influence on the deformation of a solid.

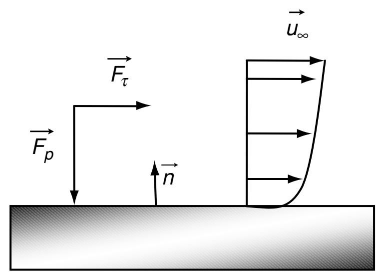

Figure 3 Sketch of the forces acting on a surface by the flow of a fluid.

$$\vec{F}_P = \int p \cdot \vec{n} \cdot dA \tag{7}$$

where by p is the pressure, A the area of the cell and n the normal vector to the surface.

$$\vec{F}_\tau = -\eta \cdot \int \nabla \vec{u} \cdot \vec{n} \cdot dA \tag{8}$$

The traction force $\vec{F}_\tau$ is defined in Eqn. (8), whereby η is the dynamic viscosity and ∇ u is the velocity gradient field. It is apparent that using only the pressure field as a boundary condition on the deformation of the solid part is not enough to achieve physically plausible results. The traction force is therefore additionally included in the force term X_b of the Fluid/Solid interface and the result is:

$$\vec{X}_b = \nabla p - \eta \cdot \nabla^2 \vec{u} \tag{9}$$

3.5. APPLICATION ON A SIMPLE FILTER MODEL

First a simple fibre model, which just consists of a number of cylinders with different thicknesses will be presented. As it can be seen in Figure (4) they are deformed due to the flow of the fluids and in dependency on their thickness. The ends of each fibre in this particular application are kept at fixed positions, which corresponds to a zero-displacement boundary condition.

In Figure (5) on the left hand side the pressure distribution on the fibres is displayed, whereas on the right hand side the traction force variation is illustrated. Red areas mark those with high values of the quantity.

It is evident that pressure and traction forces mapped on the fibres show physically plausible results. The pressure is high on the front side of the fibres whereas the traction forces show high values on the sides. Therefore their model implementation is regarded to be correct.

Corresponding to this the absolut fluid flow velocity around the fibres is displayed in Figure (6).

In this figure the regions with almost zero velocity are marked blue. As expected, they can be found in the downstream regions of the fibres.

This is also confirmed by the velocity vector field, shown in the immediate neighbourhood of the fibres, which is depicted in Figure (7).

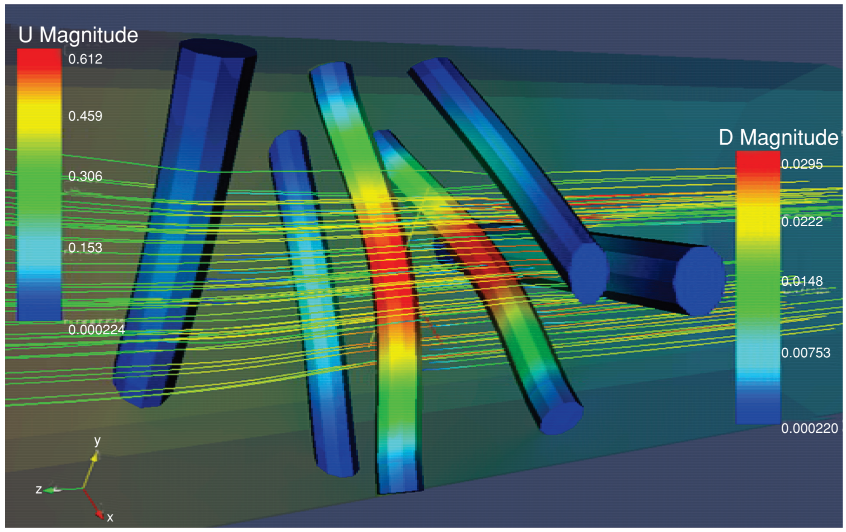

Figure 4 Model of fibres with different thickness bending in the flow of fluid.

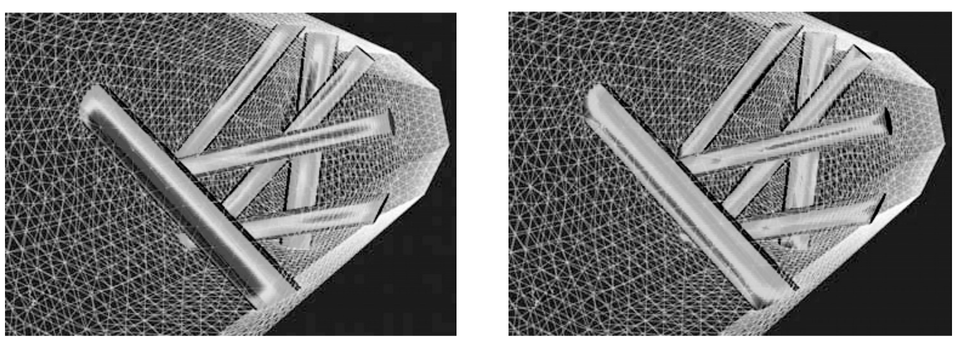

Figure 5 Pressure and traction induced by the fluid.

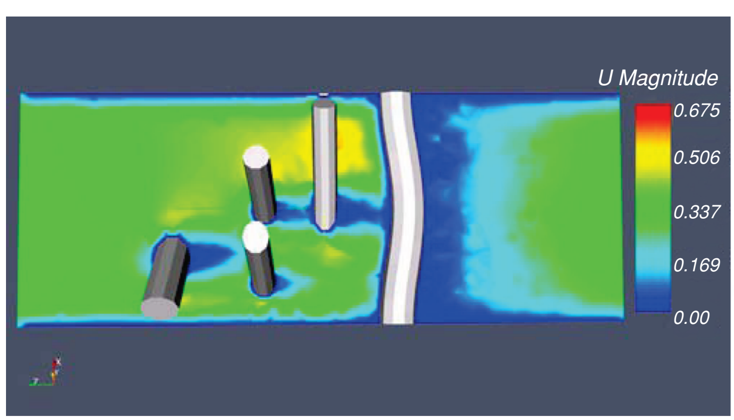

Figure 6 Velocity magnitude within surrounding fluid in a cut plane.

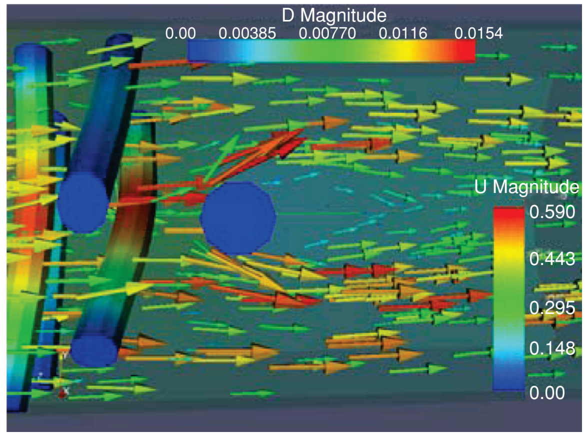

Figure 7 Velocity field of fluid is deviated around fibres.

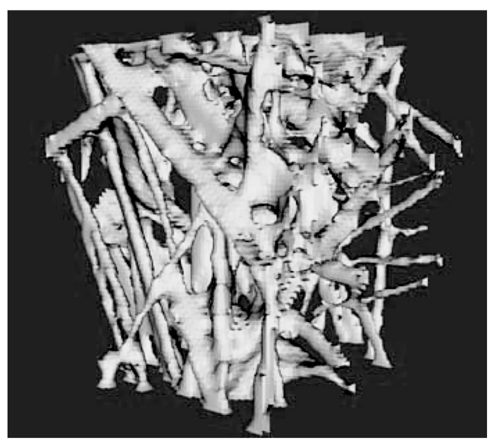

Figure 8 Reconstructed filter material, transformed into a voxel based grid.

The next example, Figure (8), shows the application of the algorithm to the behaviour of a realistic filter material sample. The geometry data was imported and reconstructed from CT-scans and transformed into a voxel based computational grid, readable by OpenFOAM [1].

In Figure (9) it can be seen that the bending of the fibres due to the flow of the oil reflects reality. The thin fibres are displaced much more, marked by red colour, than the thick and connected fibres, which are marked blue.

3.6. FURTHER IMPROVEMENT OF THE FILTER MODEL

Until now the ends of the fibres in the simulations have been fixed. This does not reflect reality. Since just a microscopic piece of the filter is modelled it means that fibres continue outside of the modelling domain. To mimic reality in a better way a new boundary condition was implemented. This implies that the fibres are not fixed anymore but rather can slide on their ends as sketched in Figure (10).

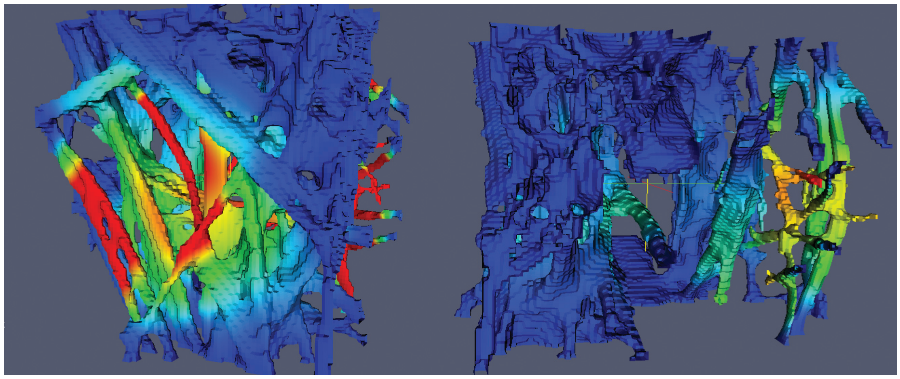

Figure 9 Fluid-structure interaction applied on a real filter material.

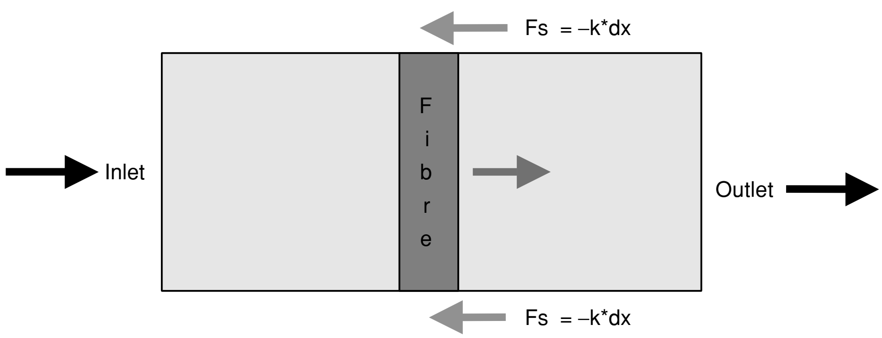

Figure 10 New boundary condition.

Each fibre can slide along the boundaries of the computational domain until a force retains it. This spring force, which acts reversely on its motion, is defined as

$$F_s = -k \cdot dx \tag{10}$$

where by k is the spring constant, which is set to an empirical value, and dx is the sliding lenght distance of the fibre along the boundary. This implies that the force increases with distance and eventually hinders the fibre to move further. An equilibrium results between the fluid forces acting on the fibre and the spring force.

During the implementation of this new boundary condition another problem occurred. After the equilibrium was reached, the fibres start to oscillate in the fluid stream, as shown for example in Figure (11). The onset of this phenomenon affected the computational stability and resulted in catastrophic programm failures. To avoid this, an improved procedure was designed, where the fibre movement is frozen, when its motion becomes small, compared to the overall displacement of the computational domain, i.e. when the equilibrium condition is reached for this particular fibre.

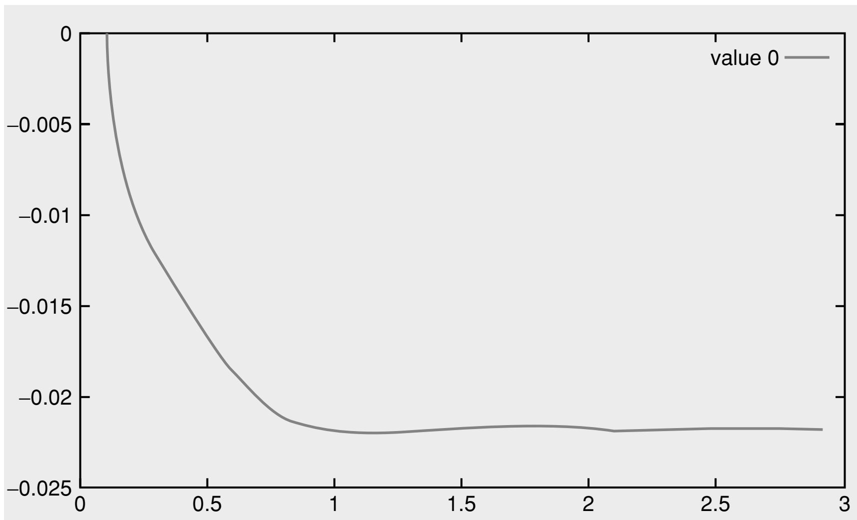

Figure 11 Oszillation of a fibre after establishing equilibrium.

During the flow of the fluid, the pressure induced forces can change. Thus the motion of fibres is not frozen for the rest of the simulation. It is possible for them to start moving again if the pressure field changes imply this.

The implementation of this new boundary condition is considered to be a major innovative step for the application of the complete mode to real geometries. Once again it should be stressed that simulations on a microscopic level are important for obtaining global material parameters, such as permeability and true surface/volume evolution. The integral results of these simulations can be used for analysis of complete filter systems, where, on a global level, porous materials are treated as homogenous regions.

4. CONCLUSION

In this paper the first part of a filtration simulation tool was developed. The program consists of an algorithm to simulate interactions of the fluid with the solid regions on the basis of the C++ based CFD platform OpenFOAM [1]. The fluid conditions, for which the applied physics are valid, are highly viscous, relatively slow, incompressible and isothermal fluid flows. This flow leads to small deformation effects of the filter fibres, for which the algorithm was developed. In both regions, fluid and solid, different equations are governing and hence have to be coupled. This coupling is handled by the pressure force and shear force interaction terms of NAVIER-STOKES equations for the fluid phase and HOOK's equations for the solid phase. The underlying computational mesh is adjusted to the deformation of the solid region during each single time step. The main advantage of this algorithm is that all of the steps listed above are realised by one single solver. Since OpenFOAM features a strictly modular programming structure, the stand alone development of a fluid-structure interaction solver was possible. Considering the limitations cited above, simulation of filtration processes using any kind of hydraulic oil and filter fibre material are possible. Current development efforts on this subject aim towards increasing the range of applicability of the solver. The main goals are to be able to simulate more significant fibre deformation effects, investigate the influence of filter material and thus the different deformation characteristics on the overall oilfilter.

The next task is to develop the second part of the filtration simulation tool, which is the filtration of particles from the flow of the fluid [13]. The combination of both will serve as a tool to simulate the filtration processes in every possible filter application from the coffee machine to the automotive industry.

5. REFERENCES

[1] http://www.opencfd.co.uk/openfoam/

[2] http://www.gnu.org/licenses/gpl.html

[3] M. Mataln, W. Brandstätter, A Unified Approach to Model Fluid-Structure Interactions, Universitaet Leoben (2004)

[4] Wilhelm Brandstätter, Flow and Combustion Modelling, lecture notes, Montanuniversitaet Leoben, Austria (2004/05)

[5] F. D. Fischer, Strength of Materials, lecture notes, Montanuniversitaet Leoben, Austria (2002)

[6] F. D. Fischer, Fluid Mechanics, lecture notes, Montanuniversitaet Leoben, Austria (2002)

[7] Comet Version 2.00, User Manual, Institut of Computational Continuum Mechanics GmbH, (2002)

[8] W. Bohl, Technische Strömungslehre, (2002)

[9] Hrvoje Jasak & Željko Tukovič, Automatic Mesh Motion For The Unstructured Finite Volume Method, ISSN 1333–1124 (2007)

[10] William H. Press, Saul A. Teukolsky, William T. Vetterling, Brian P. Flannery, Numerical Recipies in C, Second Edition (1992)

[11] Bjarne Stroustrup, The C++ Programming Language, Special Edition (2000)

[12] http://www.cfd-online.com/Wiki/CFD-Wiki:Introduction

[13] G.Boiger, M. Mataln, B. Gschaider, W. Brandstätter, *Simulation of Particle Filtration Processes in Deformable Media, Part 2: Large Particle Modelling*, ICE Strömungsforschung GmbH, Montanuniversitaet Leoben, (2007)