International Symposium
On Turbulence and Shear Flow
Phenomena (TSFP-8)
August 28 - 30, 2013 Poitiers, France

NUMERICAL MODELING OF FOULING PROCESSES ON STRUCTURED SURFACES

J. Klunker*, J.Turnow, N. Kornev
Institute of Modeling and Simulation
University of Rostock
Albert-Einstein-Str. 2, 18059 Germany
*jens.klunker2@uni-rostock.de

ABSTRACT

Concerning the operation of a heat exchanger, there are many problems that decrease the efficiency and result high economic costs (Petermeier (2003)). One of the most serious problems is the deposition of undesired particles, which is called particle fouling. The main disadvantages are a higher pressure loss along the heat exchanger and the reduction of heat transfer. Therefore, it is important to consider fouling in the designing process. This paper deals with the capability of numerical simulations to investigate and predict the growing of a fouling layer in heat exchangers. Regarding this aim, a new numerical method was developed and used for the investigation of structured surfaces. The main challenge in numerical simulations is to reduce the calculation time with a simultaneous high accuracy and representation of the physical behaviour. This paper introduce a new numerical method based on particle motion which is able to simulate the growing of a fouling layer caused by particle fouling as well as it consider the flow next to the fouling layer and the heat transfer through the fouling and liquid phase. Furthermore, the presented model depends on less constants than other common models with a simultaneous bigger range of functionality. Additionally, the used constants are not depending on the geometry, only on the matter properties.

Governing Equations

I. Fluid Equations

The fluid phase is treated by the Navier-Stokes-Equations (1) and the turbulence can be calculated by URANS or LES. To satisfy the continuity equation (2) the PIMPLE-pressure-correction is used. Further assumptions are an incompressible flow and a newtonian fluid.

$$\frac{\partial \rho \vec{u}}{\partial t} + \nabla \cdot (\rho \vec{u}\vec{u}) = -\nabla p + \nabla \cdot \mu \nabla \vec{u} - \alpha \frac{\mu}{K} \vec{u} \tag{1}$$

$$\nabla \cdot \vec{u} = 0 \tag{2}$$

where $\vec{u}$ is the fluid velocity vector, ∇p the kinematic pressure gradient, μ the kinematic fluid viscosity, α the phase fraction and K the permeability coefficient. The additional term $\alpha \frac{\mu}{K} \vec{u}$ represents the additional drag caused by the fouling phase, also known as Darcy-term.

To consider the heat flux through the solid and fluid phase the temperature is handled by a passive scalar transport equation (3).

$$\frac{\partial \left(\rho(\alpha) c_p(\alpha) T\right)}{\partial t} + \nabla \cdot \left(\rho(\alpha) \vec{u} c_p(\alpha) T\right) = \nabla \cdot \lambda(\alpha) \nabla T \tag{3}$$

The different phases are represented by the phase fraction α, which is defined by

$$\alpha = \frac{V_{phase}}{V_{cell}}. \tag{4}$$

II. Particle Motion

A high accuracy regarding to the fouling mechanism, is realised by the Lagrange Particle Tracking (LPT) (5) (Andric, J. (2009)).

$$m_p \frac{d\vec{u}_p}{dt} = F_w + F_g \tag{5}$$

In this paper, for the shown test cases the particles are influenced by the drag

$$F_w = \frac{3}{4} \cdot \frac{c_w Re_p m_p \nu_c}{\rho_p d_p} \tag{6}$$

and gravity force

$$F_g = m_p \cdot \vec{g} \cdot \left(1 - \frac{\rho_c}{\rho_p}\right). \tag{7}$$

The interactions between particles/particles and particles/wall are described by the spring-slider-dashpot-model (see Fig. 1). In case if the interaction partner is a wall, the mass and the radius of B is set to infinity.

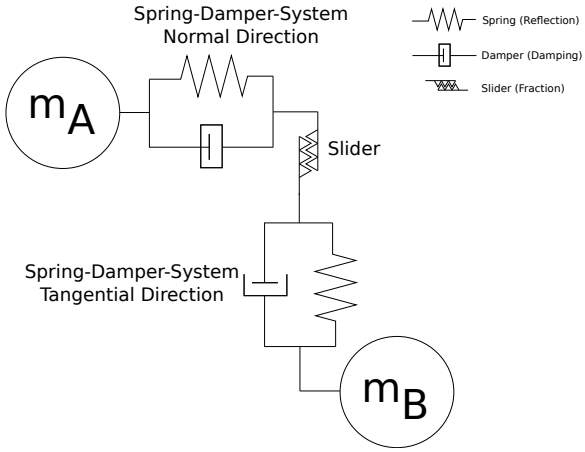

Figure 1. Spring-Slider-Dashpot-System

The normal component of the acting force, caused by the interaction, is determined by

$$\vec{F}_n = \vec{n} \cdot \left[k_n \delta^{\frac{3}{2}} - \eta_n U_{relN} \right], \tag{8}$$

where k_n is the Hertzian spring stiffness, δ the overlapping factor, η_n the damping constant and U_{relN} the relative normal particle velocity. Finally, the tangential component is calculated by the friction law

$$|\vec{F}_t| \leq -\mu \cdot |\vec{F}_n|, \tag{9}$$

where μ is the friction coefficient. The main advantage of the presented method, compared to other numerical methods, is a low calculation time despite additional functions.

Numerical Procedure

The numerical procedure is a combination of the NSE and the LPT. The two different methods are coupled by a Particle-Phase-Conversion-Algorithm (PPCA). The further phase treatment is considered by an additional source term, which is implemented in the NSE. The PPCA is illustrated in Fig. 2 and Fig. 3. The algorithm can be divided into five steps. First, every particle velocity is compared with a limited velocity which can be freely set. If the particle velocity is fall below the limit, the conversion algorithm is activated. In order to prevent a conversion of slow particles far away from the wall or a existing fouling phase, the actual cell distance to the wall and the fouling layer is checked.

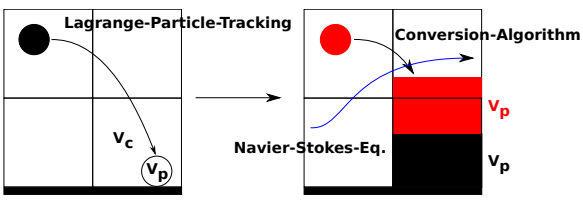

Figure 2. Concept: Particle-Phase-Conversion

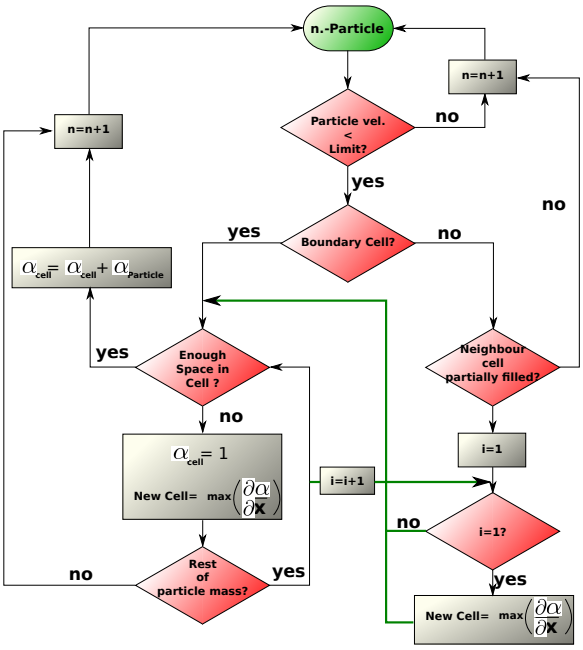

Figure 3. Particle conversion algorithm

If the boundary cell condition is satisfied the actual free cell space is evaluated and the new phase fraction is determined by the ratio of the particle volume with respect to the cell volume

$$\alpha_{cell} = \alpha_{cell} + \frac{V_{particle}}{V_{cell}}. \tag{10}$$

In case the particle mass can not be introduced into the cell - not enough space for the particle mass - a new cell is evaluated by the maximum of the phase fraction gradient of the neighbour cells. If the actual cell is not a boundary cell, the algorithm is verifying that neighbour cells are partially or full filled by particles. This condition ensure a connected deposed matter. The final step is the calculation of the released mass. Released particles and their mass are identified by the shear stresses, that acting on the fouling phase,

$$\alpha_{released} = \frac{n_p \cdot V_p}{\tau_{rel}} \cdot \frac{|\tau_c|}{V_c}. \tag{11}$$

where the released phase fraction is determined by the number of particles n_p with a specified Volume V_p in respect to a set shear stress τ_{rel} and multiplied by the actual cell shear stress τ_c divided by the cell volume V_c. This linear approach is adapted from the Kern and Seaton model (Mueller-Steinhagen, H. (2010)).

To consider the fouling layer within the Navier-Stokes-Equations (1) an additional source term is introduced to them. The mentioned term is commonly known as Darcy term, which handle the flow through a porous media. The presented conversion algorithm results an enormously reduction of calculation capacities with an increasing number of particles with respect to the single LPT calculations. A further advantage of this method, the heat transfer through different phases can be considered by a passive scalar transport equation. The only condition is the dependency of the

different terms on the phase fraction α. Furthermore, this numerical procedure is capable to deal with high density changes between the different phases. Until the particles are converted, the method realises a realistic particle behaviour by the LPT. It considers the acting forces on particles caused by the fluid and the collisions between the particles. In Addition, there is no need for a explicit (empirical) fouling model, because a natural equilibrium for the fouling layer will be reached by the treatment of the fouling layer that are included in the NSE. The only disadvantage, in respect to other multiphase methods, is the neglected transport equation for the phase fraction, because the fouling layer is treated as a wall.

The whole numerical procedure is shown in Fig. 4 as flow chart.

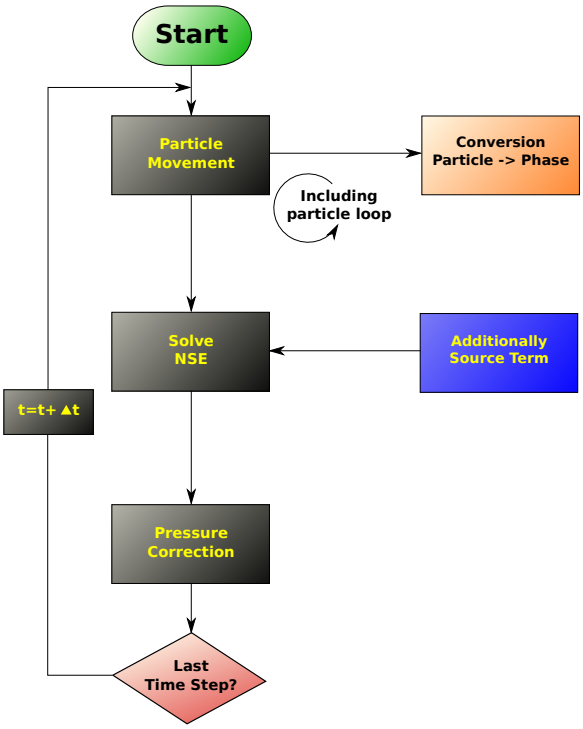

Figure 4. Flow chart of the numerical procedure

Test Cases

To validate the numerical method three different cases have been calculated with different aims:

a) Validation of the source term,
b) the Heat Transfer through different phases and
c) the conversion algorithm.

a) Validation of the NSE Source Term

For the validation of the additional NSE source term the setup includes two different simulations of a channel. They only differ in the boundary condition of the lower wall. As it can be seen in Fig. 5 the first case includes a lower wall with a common wall boundary condition. In the second case, the wall is represented by a second phase with a high viscosity.

The inlet velocity boundary condition is set to periodic and in respect a Reynolds number of 6000. The conversion

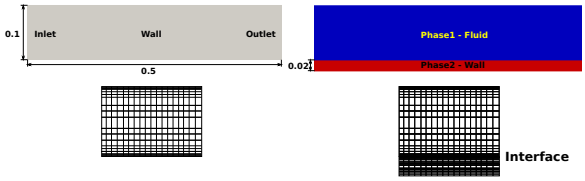

Figure 5. Test Case: Validation NSE Source Term

algorithm was completely deactivated. The turbulence was treated by the LES-method and the Mixed-Smagorinsky-Model.

From Fig. 6 it can be deducted the different boundary conditions show only a small difference along the fluid-solid-interface. The small difference is neglectable as long as the resolution of the interface is good enough. It has been shown by the analysis of the friction coefficient that the implemented source term is working properly. The difference between the two different simulations is lower than 5%. The presented method predict the influence of the deposed matter properly. Furthermore, the introduced source term results a no-slip condition as a normal wall boundary condition.

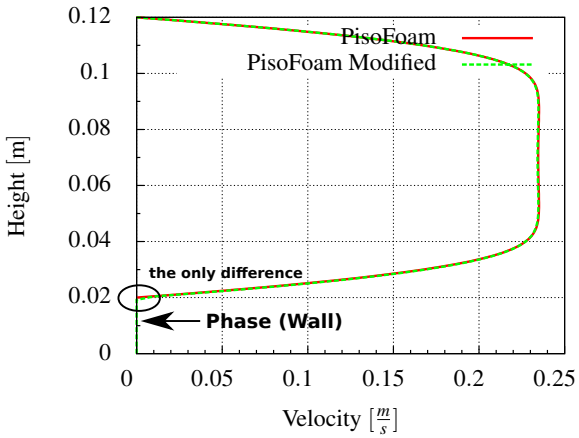

Figure 6. Test Case: Validation of the NSE Source Term

b) Validation of the Heat Transfer

The energy transfer equation was validated by a calculation of the contact temperature of two connected materials. Fig. 7 gives an overview about the parameters that were set. The one material is brass with a density of 8400 $\frac{kg}{m^3}$, a conduction coefficient of 92 $\frac{kg \cdot m}{s^3 \cdot K}$ and a heat capacity of 390 $\frac{m^2}{s^2 \cdot K}$. The attached material is steel with a density of 7800 $\frac{kg}{m^3}$, a conduction coefficient of 450 $\frac{kg \cdot m}{s^3 \cdot K}$ and a heat capacity of 13.338 $\frac{m^2}{s^2 \cdot K}$. The heat can only be transferred by conduction where the convection is neglected.

The passive heat transfer through the different materials is shown in Fig. 8. The significant point is the interface between the attached materials. The contact temperature is correct calculated, compared with an analytical solution (see Baehr (2006)).

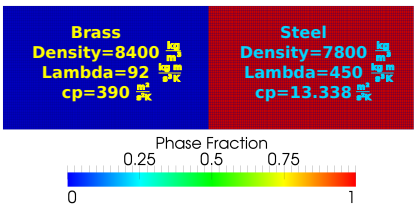

Figure 7. Test Case: Validation of the Heat Transport Equation

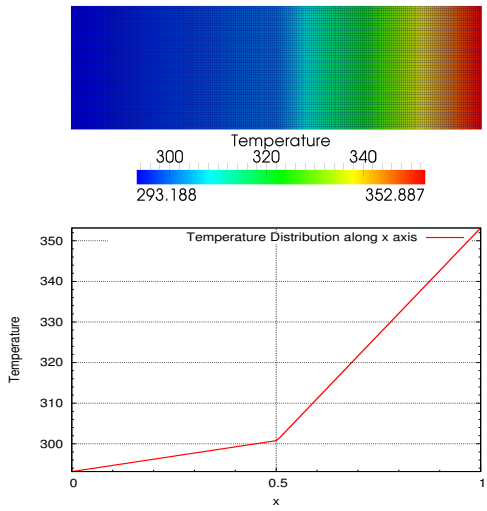

Figure 8. Test Case: Validation of the Heat Transport Equation

c) Validation of the Conversion Algorithm

For the validation of the conversion algorithm the calculated geometry is shown in Figure 9. Additionally, the table 1 displays the grid dimensions and the parameters that have been set. As it can be seen, the turbulent flow (Re = 6000) is resolved by the LES method. The chosen particles have high Stokes number of 58. It results a high response time τ_{resp} (see eq. 12) and a slow interaction between the particles and the flow.

The response time is defined by

$$\tau_{resp} = \frac{\rho_p {d_p}^2}{18\mu_f} \tag{12}$$

where ρ_p is the density of the particles, d_p the particle diameter and μ_f the dynamic viscosity of the fluid. A fast particle settling could be archieved.

The result of the conversion validation is shown in Fig. 10. Different time-steps - gap of 20 seconds - are compared regarding the height of the fouling matter. The calculations have shown a properly working conversion algorithm. As expected, the fouling threatened areas - gaps between the riblets - are good represented by the combination of the LPT, the conversion algorithm and the NSE. At the inlet, the influence of the fouling matter (brown) is shown in the enlarged section of the velocity field. It results a acceleration

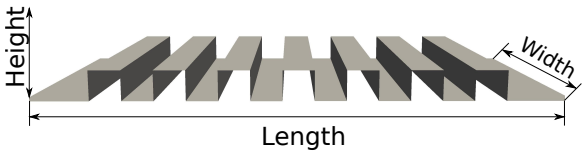

Figure 9. Grid Dimensions

Table 1. Test Case Conditions

Numerical Method	LES
Dimensions (HxLxW)[m]	0.027x0.16x0.08
Re number	6000
Riblet Height	H/3
Number of injected particles [1/s]	100000
Stokes number	58

of the velocity in the middle of the channel. Finally, almost a full filling of the gaps between the riblets could be shown.

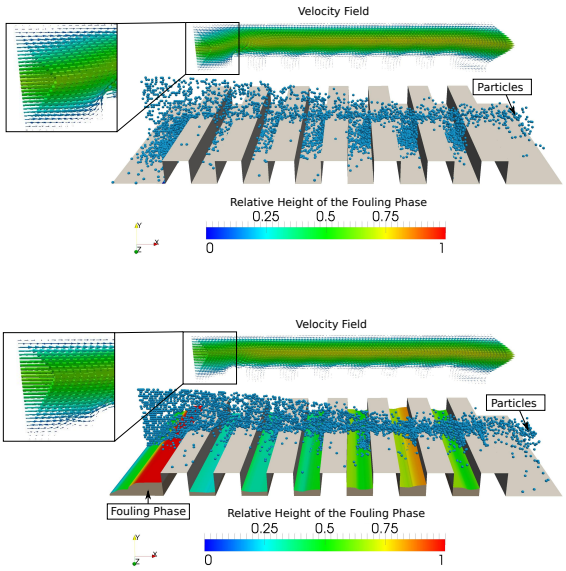

Figure 10. Test case at different times: 5s (top) and 25s (bottom)

Conclusion

In this paper a new method to predict particle fouling was illustrated. The presented method includes a combination of a Euler-Lagrange-Method. The new method and its

functions have been validated on three different cases with respect to different aims: properly working of an additional source term, heat transfer through different phases and the correct working LPT-Euler-conversion. Compared to other empirical fouling models (for example Kern and Seaton), the presented method is almost independent of experimental data and is further suitable to predict the fouling behaviour on structured surfaces.

Furthermore, a reduction of calculation time could be achieved by the particle conversion algorithm in dependence on the number of particles that have been initiated at the inlet. But even more, the flow next to the fouling layer (interface) is correct represented by the fluid phase. A further advantage is the consideration of the heat transfer. That is an important fact, because the majority of fouling mechanism are depending on the temperature. But the most valuable advantage of the presented method is the simple expandability of the new numerical fouling approach. Different fouling boundary conditions can simply be implemented.

REFERENCES

Andric, J. 2009 Lagrangian particle tracking of spheres and cylinders.

Petermeier, J. 2003 Numero-fuzzy modell zur Beschreibung des Foulings bei der Erhitzung von Milch. PhD thesis, Technische Universitaet Muenchen.

Baehr, H.D.; Stephan, K. 2006 Waerme- und Stoffuebertragung. vol. 5, Springer

Mueller-Steinhagen, H. 2010 Heat Transfer Fouling: 50 Years After the Kern and Seaton Model.Taylor & Francis