V European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2010
J. C. F. Pereira and A. Sequeira (Eds)
Lisbon, Portugal, 14–17 June 2010

SIMULATION OF THE FOULING LAYER EVOLUTION IN HEAT TRANSFER SURFACES

E. Suárez, C. Paz*, J. Porteiro, and A. Eirís

*CFD Simulation Group University of Vigo
Lagoas Marcosende 9 CP-36310 Vigo (Spain)
e-mail: cpaz@uvigo.es

Key words: CFD, Fouling thickness, Fluent, collapse cells

Abstract. *This paper presents a comprehensive numerical simulation methodology developed for studying fouling effect in heat transfer surfaces, with a particular focus on fouling layer thickness evolution.*

The main objective of this research is to develop a fouling predictive tool integrated in a commercial CFD software, FLUENT. So, in each computational cell adjacent to a wall the instantaneous fouling thickness is calculated at every time step and it is assumed that a cell is fluid until the deposit thickness clogges it, in this moment the cell is converted in solid with specific properties. As the computational strategy has not mesh or boundary restrictions, it is adaptable to any geometric configuration, as enhanced heat exchangers surfaces, easily.

The results obtained with this model are compared with experimental results obtained in a test bench, giving an acceptable level of agreement. Hence, this model represents a valuable tool for the prediction of the time evolution of the performance of heat transfer surfaces exposed to fouling.

1 INTRODUCTION

The deposition of fouling material in heat transfer processes[1,2] increases the flow pressure loss and decreases the temperature efficiency[3], meaning four money drains: The oversize device manufacture cost, the maintenance cost and the cost associated to not recovery energy and supplementary energy needed. Therefore the fouling suppose an important increase of the operating cost, it is estimated around 700 million dollars per year only in EEUU[4,5].

The research on fouling has been tackled from several perspectives, experimental research, theoretically and more recently computationally. Fouling is a fickle process, and the starting point of fouling dynamics studies is the well known Kern-Seaton approach[6], accepted by most of the research and enhanced later by Taborek et al.[7] and more recently by Webb and Kim[8]. These are based on that net grow of the fouling layer depend on the relative contribution of two simultaneous opposing processes, deposition and removal. The use of this model type represents an advance focusing in functionality and easy application for practical problems. This formulation is very simple but allows including easily the main phenomena implied in deposition and removal.

The numerical simulation of fouling has generally been directed through the use of virtual deposited mass or change the conductivity of a constant thickness wall to reproduce the fouling effect but not consider the depth evolution[9]. Hence, with this kind of approach, the effect of fouling on heat transfer is easily computed, but no modification of the flow pattern is computed and consequently pressure drop evolution and mayor modifications of the geometry (such as clogging) are impossible to estimate. For instance S. Kaer[9] proposed a virtual fouling thickness in an ash deposition studied.

The object of the current research was to develop an innovative computational algorithm designed specifically for numerical implementation of fouling models. This tool has been implemented and coupled by means of User Defined Functions (UDF) with the commercial CFD software package ANSYS-FLUENT 6.3.26.

The comparison of the predicted time-evolution of pressure drop and effectiveness of EGR coolers exposed to fouling from real diesel exhaust gases will serve as the validation of the proposed model. However, thanks to its simplicity and adaptability, the model proposed can be employed as a general framework to test and validate different formulations of the each of the processes involved in the fouling of any heat transfer surface of component.

2 FOULING MODEL

The soot layer means and additional thickness for the heat transfer surfaces and create an insulating layer of particulate matter and hydrocarbons which is translated into an additional resistance to the heat dissipation. This additional resistance is included in the calculation of thermal performance using the fouling factor, defined as the increase of thermal resistance of a heat exchange surface.

The growth of the layer is not constant, but is a rate-dependent phenomenon. Experimental and previous computational results show that exhaust gases fouling process follow an asymptotic approach with a good agreement. The temporal evolution of fouling process can be summarized in three basics steps. The process starts without any deposited matter, in the early stages of the fouling only a small portion of the surface is covered with deposits, and with time the deposits start building up until the asymptotic conditions, following the curve shown in Figure 1. The asymptotic hypothesis was easily proved adjusting experimental data to an exponential rise to maximum expression which a good agreement was obtained[10].

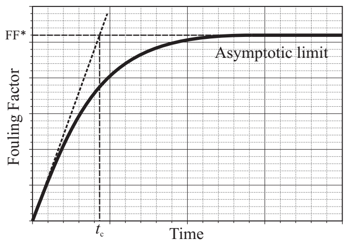

Figure 1: Asymptotic fouling process.

In the order to validate the proposed computational model, a specific physical model should be implemented. Most of bibliography fouling studies, assume that deposit accumulation is the result of two simultaneous opposing events, and the net fouling rate is the difference between deposition and the removal rate. Eq.1.

$$\dot{m}_f = \frac{dm_f}{dt} = \underbrace{S_d \cdot \left(u_d + u_{th}\right) \cdot C_b}_{Deposited\ rate} - \underbrace{\frac{\tau_w \cdot x_f \cdot \rho_f}{\xi}}_{Removal\ rate} \tag{1}$$

The variables that influence in both mechanisms have intricate interactions on a local scale by the nature of fouling. Deliberately, the model can embrace different fouling mechanisms providing the fouling is asymptotic: particulate fouling[11], diffusion, hydrocarbons condensation, thermophoresis[12], and precipitation.

In the above expression adopted for deposition and removal some variables can be obtained from correlations, while the other must be derived from experimental results. The bulk particulate concentration C_b and fouling density ρ_f are constants whose values can be estimated from experimental results[13]; u_d, is the velocity deposition taken of Wood correlation[14] and is u_{th}, the thermophoretic velocity, enhances the particle transport from hot to cold areas, is taken from Talbot et al.[15], which several experimental studies suggested that is accurate enough[16].

The removal rate is assumed to be directly proportional to the thickness of the fouling layer x_f, and local wall shear stress τ_ω calculated by the CFD program, and inversely proportional to the strength both factor ξ.

A one layer model was considered in this work, therefore the fouling layer has uniform properties. Hence, the model developed is a semi empirical and phenomenological model that allows the prediction of the fouling layer evolution based on two parameters, the sticking probability S_d and ξ, associated to deposition and removal respectively, which will be estimated from experimental data.

3 NUMERICAL MODEL

The local geometric details like tube ribs would change the mean flow conditions affecting indirectly to the fouling process itself. Thus to the requirements previously defined, simple, low computational cost and accuracy, it was also important for the fouling model implementation to have in mind that local effects should be captured.

Different numerical fouling approaches, based on a moving boundary concept, fill and transform fluid cells by the reconstruction of fouled profile, such as the work of El-Batsh[17] to generate the fouled profile of turbine blade surfaces.

To our knowledge, the model presented in this paper provide the first CFD fouling model taking into account temporal and locally evolution of thickness, and this is probably the key of this study.

In each computational cell the instantaneous fouling thickness corresponding to the volume of the deposited particles was calculated in every time step. The volume of each fluid cell adjacent to a wall was compared to the fouling thickness and it was assumed that a cell is fluid until the thickness fill to the top of the cell.

When the fouling thickness was larger than the fluid cell, the fluid cell was converted externally in solid with specific fouling properties. This simplification is possible because the height of the cell region that might be fouled was low enough. In our case the height of cells closer to the walls were in the order of 10 μm. This fouling growth procedure until the filling up of the cell is schematically shown in figure 2. The height of cell has been intentionally altered for an easier understanding of the process.

For this purpose, a collection of User Defined Functions, (UDF) and batch process subroutines were created to obtain the volume of each cell, calculate fouling thickness during the time-step and if necessary to perform all the operations needed for changing the type of the cell.

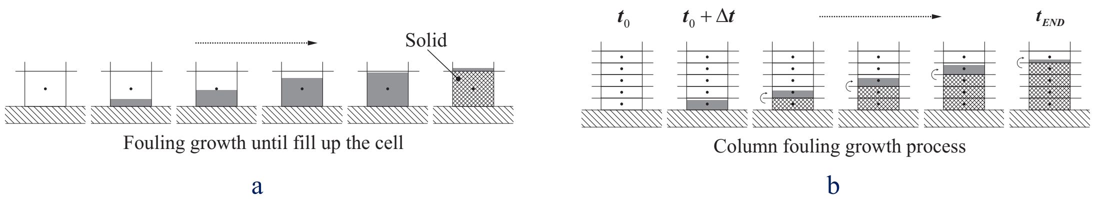

Figure 2: Temporal evolution of fouling thickness by filling fluid cells.

In order to obtain a continuous profile the average deposition in each column was interpolated with its neighbors. The average deposition was used to calculate the smoother fouled contour.

To reach the asymptotic values of the fouling process, a transient simulation must be computed until steady performance is obtained, therefore the numerical methodology was coupled to the flow solution by means of a sequential loop.

The process starts calculating under clean conditions and is solved by Fluent. The solid phase is calculated externally with the fouling model, and using CFD flow parameters it was estimated the temporal variation of fouling thickness. The mesh is updated and finally the loop is repeated until the asymptotic stage.

The fouling proposed model implies an increase in the mesh requirements, in the same order of magnitude than the use of an enhanced wall treatment turbulent model.

To solve the near-wall viscous region and apply the fouling model in adjacent wall cells, two regions were done in boundary layer, the near-wall region, with 30 fine and uniform prismatic layer, with a size adjusted to obtain a y^+ value around 1, and 15 prismatic layers with linear growth were placed in the second region, in order to achieve a smoother transition between the prisms and tetrahedral cells.

The RANS-based modelling approach and the segregated solver were adopted. The viscosity-affected near-wall region was resolved by enhanced near-wall treatment using the standard k-epsilon two-layer model of Wolfshtein incorporated in the Fluent code[18]. Turbulent heat transfer is modelled using the concept of Reynolds' analogy to turbulent momentum transfer. The solver and main properties are shown in table 1.

Solver		Properties	
Space	3D	Density	Incompressible ideal gas
Solver	Segregated	c_p	Polynomial: 960.05 01526 -3e-5
Formulation	Implicit	Thermal Conductivity	Polynomial: 0.0131 5e-5 -3e-9
Time	Steady	Viscosity	Sutherland Law: 1.716e-5 273.11 110.56

Table 1: Solver and properties of simulated exhaust gases.

4 RESULTS

When this method was tested, a converged solution for each transitory fouled situation was obtained, and the results obtained fit with the experimental values. A scheme of experimental versus model results is shown in figure 3.

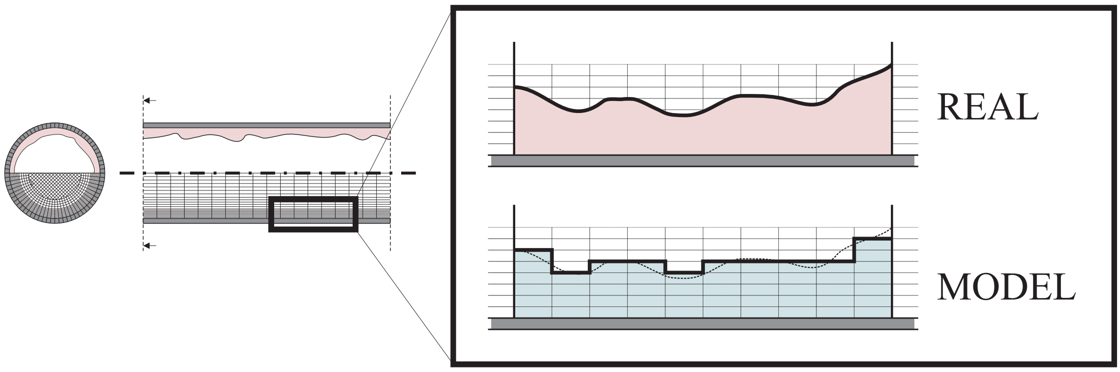

Figure 3: Scheme real vs model profile fouling.

The study has been particularized to fouling deposition in exhaust gases heat exchangers, shell and tube type. In order to carry out the simulation with the available computational resources some simplicity hypothesis were introduced: the coolant circulation through the shell side is replaced by a thermal boundary condition (h_{ext}, T_{ext}) validated by previous simulations; and an equal distribution of flow in each tube of the bundle is assumed.

To ensure the equilibrium beside accuracy and computational effort in the cleaning conditions simulation, given previous experiences in similar problems simulated[19] it was decided to reduce the interest area to the main heat exchanger component in the EGR cooler, a single tube.

The evolution of fouling factor predicted by CFD model has been compared with experimental fouling factor, acquired from bench test facility[13]. The CFD predicted values under the same conditions show good agreement with experimental data in terms of local fouling behavior and rates. In Figure 4 different CFD predicted and experimental fouling factors are shown.

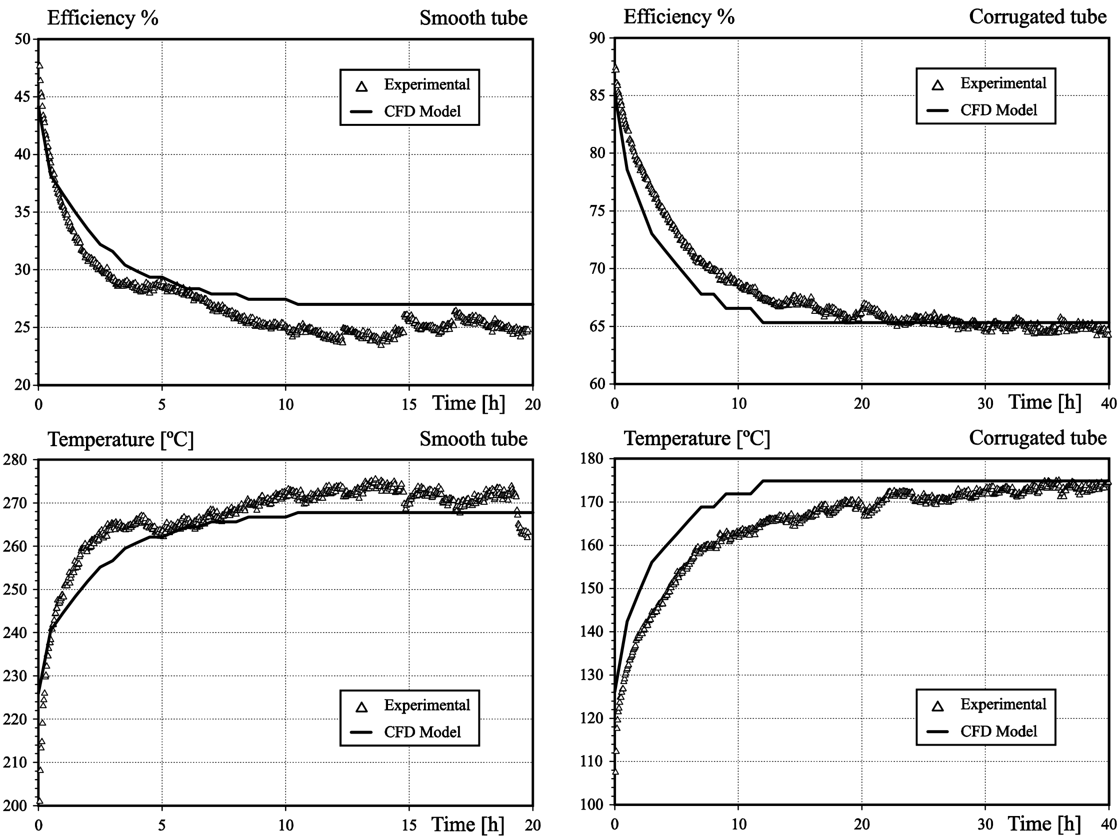

Figure 4: Real vs model thermal efficiency and temperature evolution in smooth and corrugated tube.

In order to study methodology flexibility of the proposal model, different geometries heat exchangers were tested, helical rib tubes, circular, squared and rectangular and using several work conditions.

The proposed technology was applied to fouling prediction in Diesel exhaust gases heat exchanger tubes. The standard working conditions of mass flows, soot particle concentration and size[20] were used. An example of fouling results obtained in this study is shown in figure 5. The biggest fouling thickness was located in minimum wall shear stress areas, according with early experimental research[21].

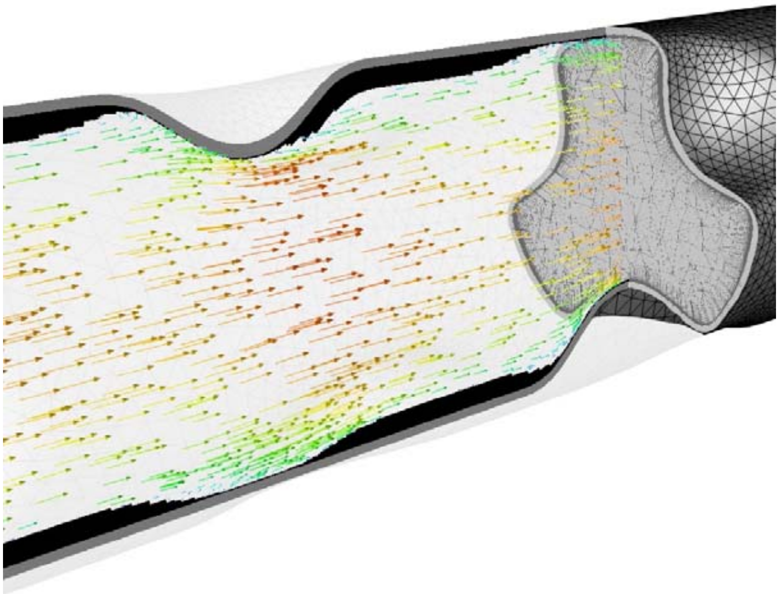

Figure 5: Internal fouling thickness layer into an enhanced heat exchanger tube.

5 CONCLUSIONS

The CFD model results have shown good agreement with experimental test, allowing the validation of proposed methodology. The versatility of the model is proved, simulating completely different geometries, has been proved in smooth and corrugated pipes, square sections tubes, plate heat exchangers, without remarkable problems, so this model is applicable to any geometry. The methodology computational cost is assumable, another important initial requirement.

The fouling physical model implementation into the computational model was maintained open to be interchangeable, which allow easily test different models under the same computational platform. The detail provided by the CFD fouling model demonstrates that scale growth has a considerable impact on the hydrodynamics of the system, and vice-versa.

As final conclusion, this model would represent a valuable tool for the prediction of the main aspects of the performance of heat exchangers exposed to fouling, and very useful to compare different previous fouling models.

REFERENCES

[1] T. R. Bott, Fouling of Heat Exchangers. Elsevier Ltd. 522 (1995)

[2] A. E. Bergles and E. F. Somerscales. The effect of fouling on enhanced heat transfer equipment. J. Enhanced Heat Transfer. **2**, 157-166 (1995)

[3] N. Epstein. Elements of particle deposition onto nonporous solid surfaces parallel to suspension flows. Experimental Thermal and Fluid Science. **14**, 323-334 (1997).

[4] A. M. Pritchard. The economics of fouling. Fouling Science and Technology, 31-43 (1988).

[5] M. Goyhenetche. Diagnostic technique et economique de l'encrassement des equipements de transfert thermique dans l'industrie fran caise. Technical report, Michael Goyhenetche Consultants, France (1991).

[6] D. Q. Kern and R.E. Seaton. A theoretical analysis of thermal surface fouling. British Chemical Engineering, **4**, 258-262 (1959).

[7] J. Taborek, T. Aoki, R. B. Ritter, J. W. Palen, and J. G. Knudsen. Fouling: The major unsolver problem in heat transfer. American Institute of Chemical Engineers, New York, 68:59-67 (1971).

[8] R. L. Webb and N. H. Kim. Principles of Enhanced Heat Transfer. Taylor and Francis (2004).

[9] S. K. Kaer, Numerical investigation of ash deposition in straw-fired boilers Institute of Energy Technology Aalborg University, Denmark (2001).

[10] T. R. Bott and C. R. Bemrose. Fouling on the gas-side of finned tube heat-exchangers. Journal of Heat Transfer, 178-183 (1983).

[11] J. M. Grillot and G. Icart. Fouling of a cylindrical probe and a finned tube bundle in a diesel exhaust environment. Experimental Thermal and Fluid Science, **14**(4), 442-454 (1997).

[12] M. C. Paz, E. Suárez, J. Porteiro and A. Eirís. Particle Deposition Computational Model for Diesel Exhaust Systems. Proceedings 10th Conference on Energy for a Clean Environment p-25 (2009).

[13] M. C. Paz, E. Suárez, A. Eirís, and C. Castaño. Experimental set up for the determination of fouling behavior in diesel engine exhausts gas recirculation systems. In Proceedings 10th Conference on Energy for a Clean Environment p-106 (2009).

[14] N. B. Wood. A simple method for the calculation of turbulent deposition to smooth and rough surfaces. Journal of Aerosol Science, **12**(3):275-290 (1981).

[15] L. Talbot, R.K. Cheng, R.W. Schefer, and D.R. Willis. Thermophoresis of particles in a heated boundary layer. Journal of Fluid Mechanics, **4**:737-758 (1980).

[16] C. J. Tsai, and H. C. Lu, Design and Evaluation of a Plate to Plate Thermophoretic Precipitator Aerosol Science and Technology, **22**:2, 172-180 (1995).

[17] El-Batsh, H. Modeling Particle Deposition on Compressor and Turbine Blade Surfaces Vienna University of Technology (2001).

[18] ANSYS-Fluent Users guide (2007).

[19] M. C. Paz, A. Eirís, E. Suárez, and C. Castaño. Heat transfer enhancement in egr coolers with internal corrugated tubes. In Proceedings 9th Conference on Energy for a Clean Environment p-87 (2007).

[20] P. Eastwood. Particulate Emissions from Vehicles. John Wiley and Sons, Ltd (2008).

[21] D. Bouris, E. Konstantinidis, S. Balabani, D. Castiglia, and G. Bergeles. Design of a novel, intensified heat exchanger for reduced fouling rates. International Journal of Heat and Mass Transfer, **48**:18, 3817-3832 (2005).