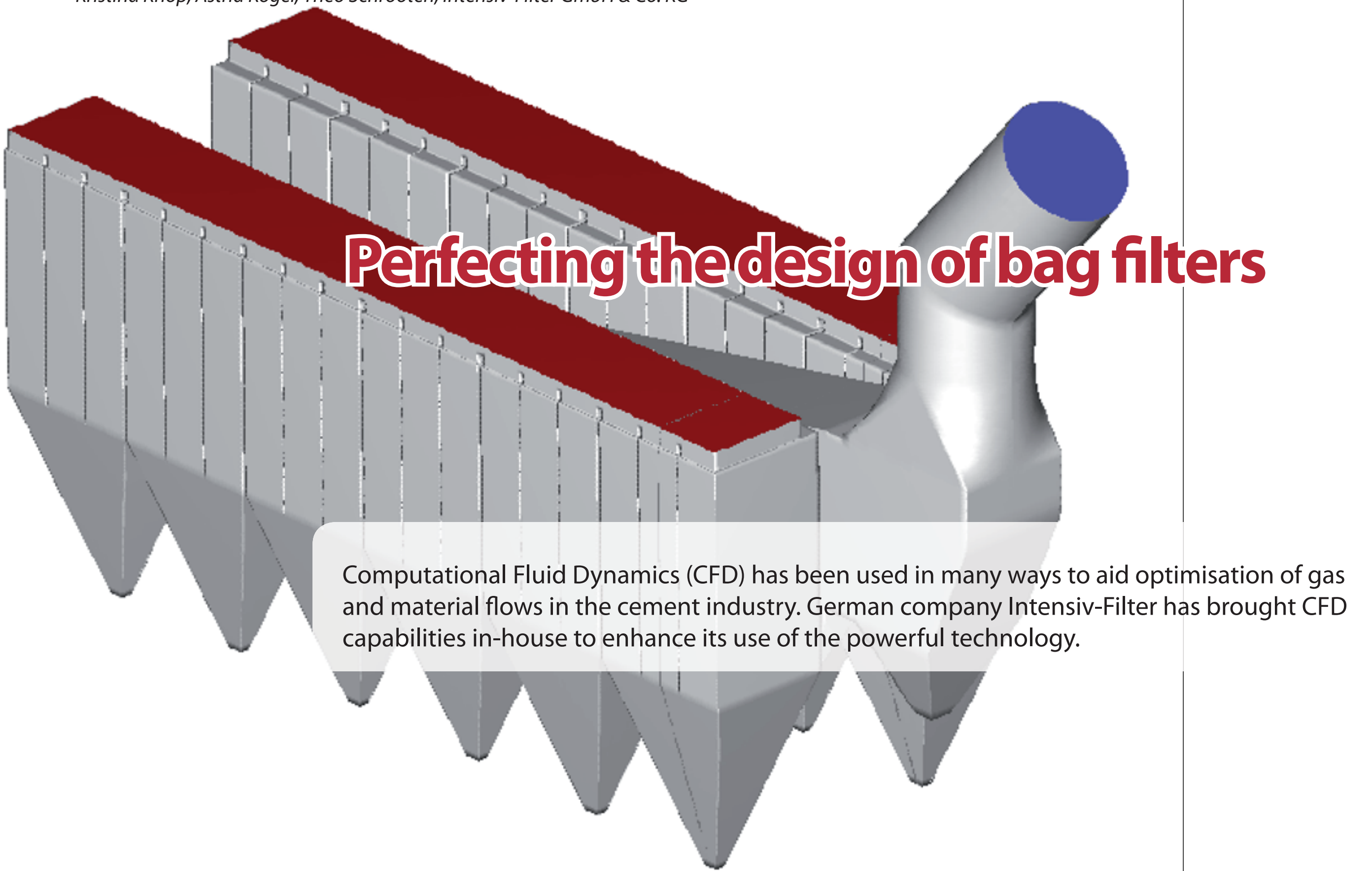
Kristina Knop, Astrid Kögel, Theo Schrooten; Intensiv-Filter GmbH & Co. KG

Perfecting the design of bag filters



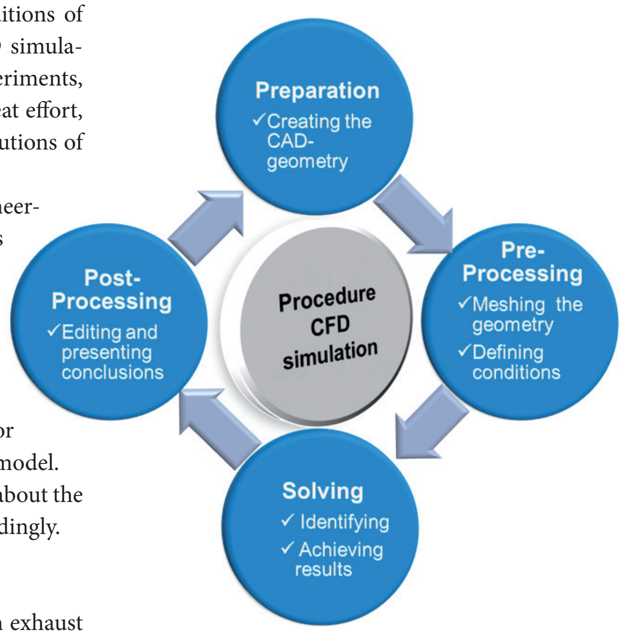
Computational Fluid Dynamics (CFD) has been used in many ways to aid optimisation of gas and material flows in the cement industry. German company Intensiv-Filter has brought CFD capabilities in-house to enhance its use of the powerful technology.

Above: CAD-generated layout of a bag house filter, ready for use in CFD computation.

Use of the 'trial and error' principle is still valid in some industries. However, the use of advanced computerised technology to aid in understanding and to minimise costs for product development and design of the plant is a much smarter method. Computational Fluid Dynamics (CFD) awakens ideas, CFD reveals, and CFD prevents mistakes - all at a much lower cost than 'trial and error.'

Modern CFD programmes permit the simulation and analysis of flows on the computer. The computer-supported analysis enables examination of the dynamics of flowing media and provides a computer model which represents the examined conditions of an installation. The special strength of CFD simulations lies in the fact that 'trial and error' experiments, which are practicable in reality only with great effort, can be limited by CFD to the most likely solutions of the problem and with a minimum of effort.

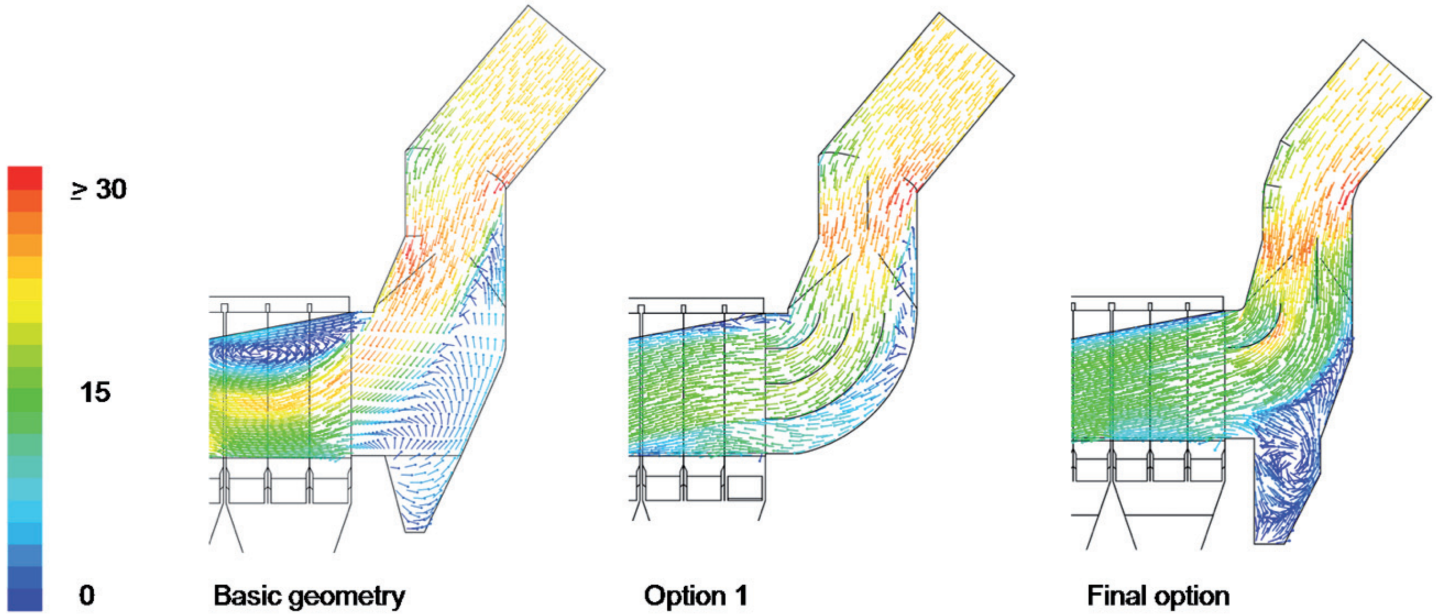
In different areas of mechanical engineering the support of development processes by computer simulation has turned out to be an effective instrument, particularly in the reduction of development costs and development time. When analysing the problem, a computer model takes the place of the original installation and experiments or constructive changes are carried out in the model. If the model is accurate enough, conclusions about the behaviour of the original can be drawn accordingly.



Right: Steps of CFD simulation.

CFD for cement industry gas cleaning

Because of the way the system operates, high exhaust gas amounts to be cleaned are typical for cement production. Therefore bag filters are basically installed for the dust removal. Only bag filters are able to keep within the increasingly stringent legal emission limits under all conditions. Beside low investment costs and maintenance costs, the filter plants must also contribute to the increase in energy efficiency. Exhaust gas amounts from 0.5-1.0million Nm^3/h are cleaned at cement mills as well as at the cooler and bypass, and volumes can be as high as 2million Nm^3/h for the kiln or raw mill dust removal. Improvements which lead to



Above: Optimising the inflow in the raw gas room and preseparator for a cement kiln or cement mill filter

a reduction of the pressure loss play an important role in this regard.

Intensiv-Filter, a leading manufacturer of filter plants for the cement industry, has made extensive use of CFD-simulations to optimise its products. In the first part of this article the basic principles of simulation studies are explained. In the second part, a number of case studies in the cement industry are discussed.

CFD-simulation studies in principle

CFD makes it possible to predict heat transfer, pressure loss behaviour, velocity profiles, particle tracking velocimetry, flow phenomena and vaporisation processes. All appropriate dimensions and physical characteristics from 'the real world' are collected in the field and a very high fidelity computer model is then constructed. A colour-coded view of the output variables of the simulation model generates a descriptive impression of the examined geometry.

In the run-up to the simulation run on the computer, the user fixes the geometry of the system. The handling of flow problems is then organised in four steps.

1. Preparation: The first step is the most labour-intensive part of the CFD process. The preparation encloses the generation of the geometry in a CAD programme. Afterwards the design of the 3-D model is imported in the CFD programme.
2. Pre-Processing: The allocation of the net (meshing) to the model is the next step. Before the numerical simulations can be started, all edge conditions must be defined first. This includes the definition of the flow inlet and outlet as well as the roughness of the pipe wall, for example. In addition, the characteristics of the flowing media to be examined must also be defined before simulation, including gas composition, temperature and pressure as well as resultant parameters like density and dynamic viscosity.
3. Solver: The necessary equation models for the system are applied in the third step, supported by sufficient computer hardware capacity for fast processing of calculations.
4. Post-Processing and analysis: The last step delivers the results of the computation and allows their final evaluation. By means of visual presentation and interpretation of the data, the flow-mechanical qualities of the installation can be divined. Two-dimensional cross-sections or three-dimensional graphics are typical.

From theoretical basics to practical uses

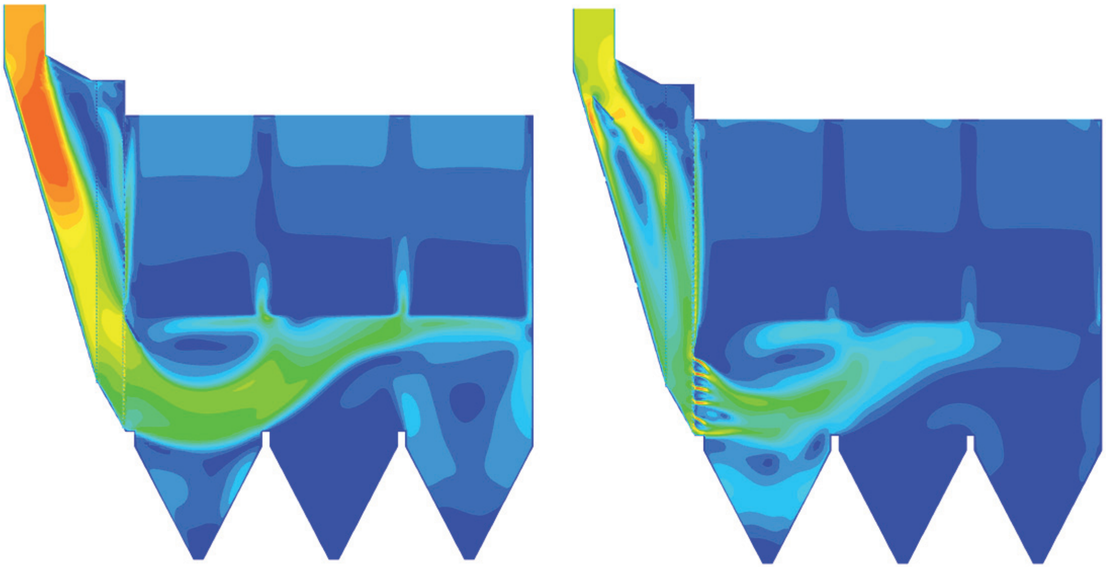
CFD makes it possible to predict system behaviour in detail, at much lower cost than testing out system configurations in the real world. Arguments for the application of computer-supported simulations are manifold:

- More exact and comprehensive analysis of the flow in plants in comparison to the use real-world 'trial and error;'
- Fast results;
- Once provided, various parameters can be varied easily;
- The effectiveness of geometry optimisation, e.g., installation of baffle plates, can be evaluated fast.

Example 1

Intensiv-Filter carries out the design of the geometry of an existing filter bag installation in the Solid Works CAD programme. A new raw gas duct was to be constructed into the filter, and the aim was to optimise the inflow of the raw gas into the filter chambers, to avoid dead zones and to reduce pressure loss.

Figure 2 illustrates the results of the CFD-simulation. The turbulence areas which unfavourably affect the flow distribution can be recognised. Different computations were carried out in the area of the filter entry and to model the flow distribution in the filter.



Above: For the retrofit of an electrostatic precipitator, the figure shows the velocity before and after optimisation with CFD.

Suitable baffle plate geometries were identified and evaluated, based on the simulation results. The finally implemented option clearly shows the effect of flow optimisations. By an optimum position of the baffle plates the recirculation zone was eliminated and an homogenised approach flow into the filter chambers was achieved.

Example 2

For dust removal for the rotary kiln at Deuna Zement GmbH, the existing electrostatic precipitator was converted to a bag filter. The raw gas room had to be equipped with filter bags in order to achieve the necessary filter surface area. The spatial restrictions and the available raw gas entry foreshadowed a disadvantageous flow distribution in the raw gas room and an unfavourable upward flow of the bag packages. The flow simulation was to provide information about the changes required in the inflow area. The aim was to provide uniform incoming flow of the bag packages by maximising the crossflow and minimising the upward flow.

Figure 4a illustrates the flow simulation before optimisation. Areas with very high velocities and turbulence areas are to be seen. A special fin plate guidance system was used, resulting in the optimisation of the inflow behaviour in the filter and filter elements.

The flow optimisation resulted in:

- Uniform inflow of the filter bags through attaining the desired crossflow;
- Uniform flow velocity in the raw gas room;
- Minimisation of the upward flow between the bags;
- Significant reduction in the filter resistance and consequent reduction of the differential pressure;
- Reduction of operating costs.

The uniformity of the bag inflow and low differential pressure in the filter has been proven in practice.

Conclusion

CFD speeds up project work in conceiving and realising industrial dust removal installation, but it also serves as a tool for basic advancements. With a suitable choice of the simulation model, optimisation possibilities close to the installation can be found. The key to efficiently solving customer-specific tasks is the networking of the CFD program with the CAD system. Nevertheless a simulation programme is only as good as the user who serves it. The model construction, the simulation realisation and evaluation need a lot of experience. The CFD experts at Intensiv-Filter have the necessary experience.

With its increased use of CFD, Intensiv-Filter has extended its core competence in the development of energy-efficient filter plants and systems engineering, from the source of emission up to the stack.

Due to the increasing importance of CFD optimisation of filter installations, Intensiv-Filter has focused on in-house use of CFD software since the beginning of 2009. CFD-based customer questions can be answered during the conceptual design of the filter plant.

Advantages of CFD	Disadvantages of CFD
Physical edge conditions (e.g., velocities, pressure) can be simulated and controlled precisely.	The model can only be an approximation of the real behaviour and can lead to faulty interpretation of the CFD results.
Specific problems can be regarded in isolation.	Net grids can be spaced too far apart.
Simulations delivers boundless data from any spatial point and from any point of time.	Computing time for very complex plant geometries is very high.
Concurrent capture of all flow parameters in one computational operation.	
Fast development of virtual prototypes permits integration of the results in early planning and development.	
Development of virtual prototypes reduces the necessary number of real prototypes or supersedes them.	
Parameter studies are possible on many models, allowing full understanding of the problem.	

Table 1: A comparison of advantages and disadvantages of CFD simulations.