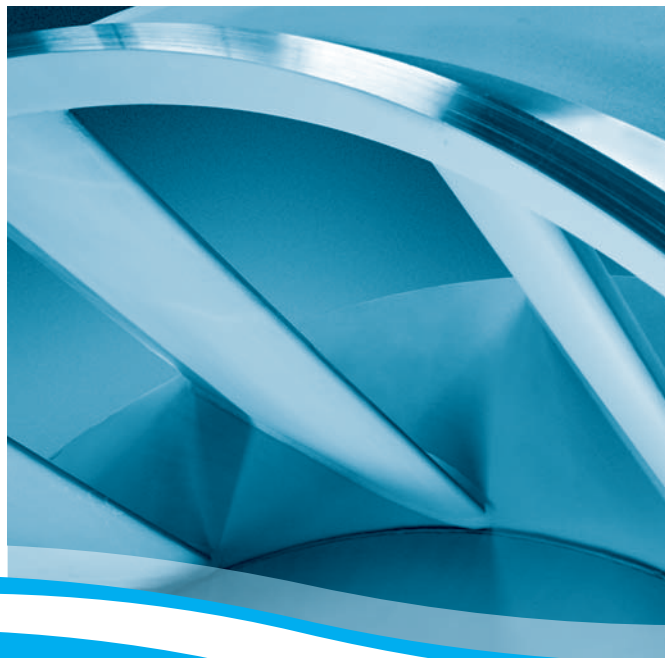
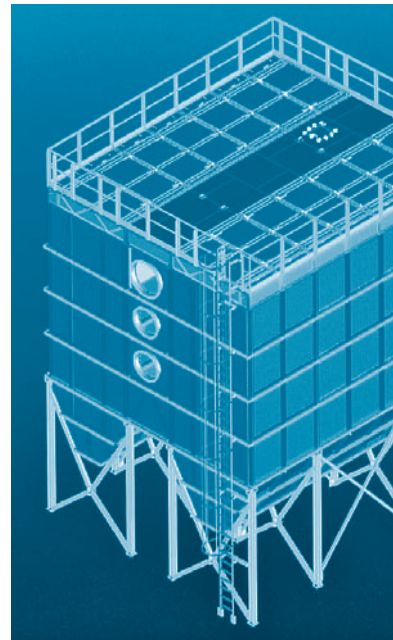
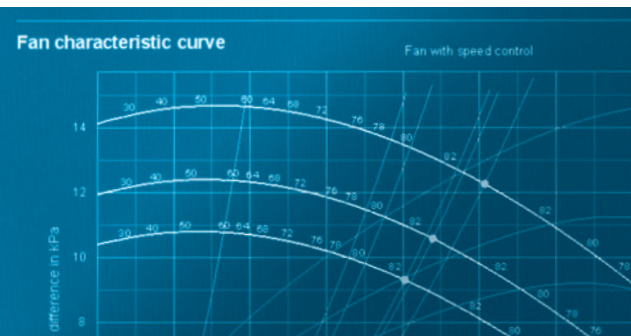
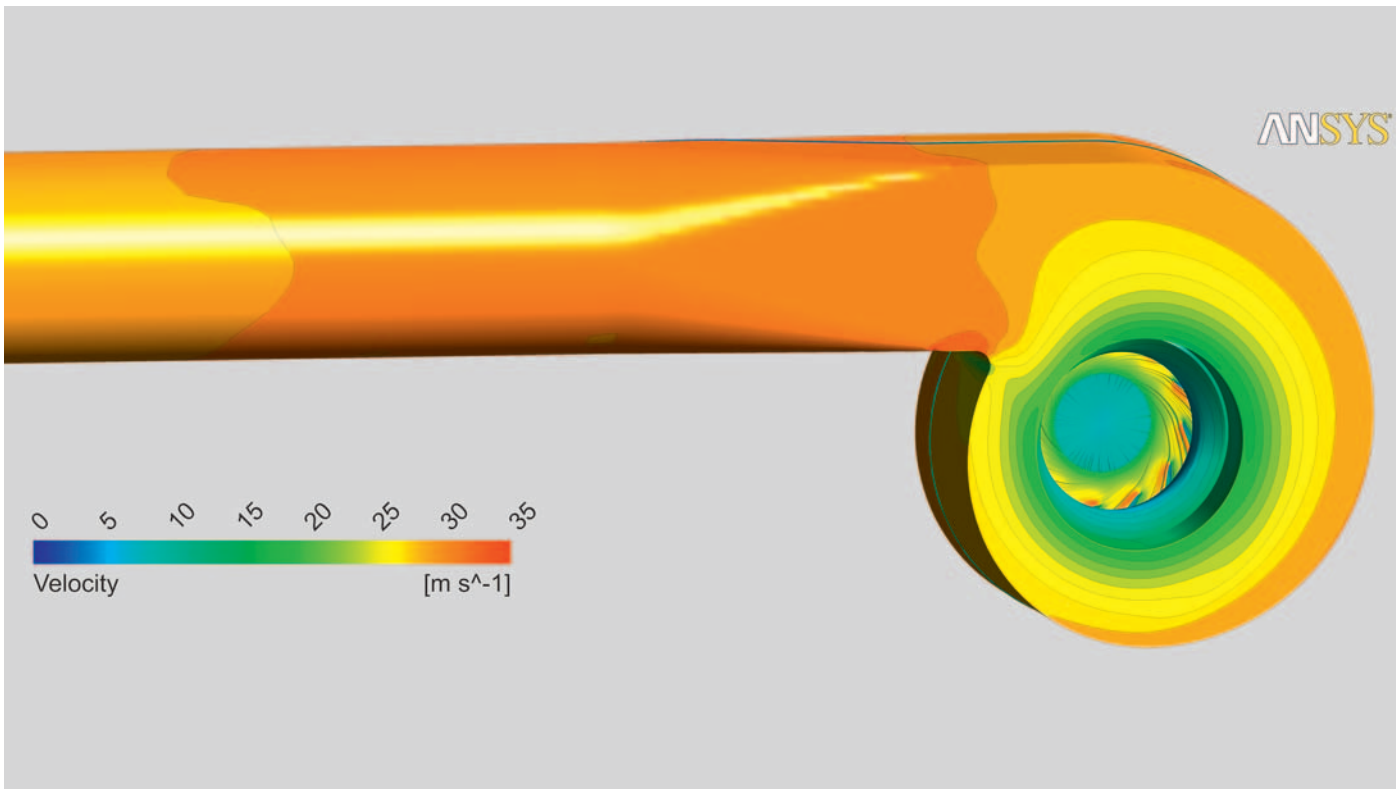


# Innovations from Venti Oelde

Computational Fluid Dynamics for fans and plants





Fan optimization with CFD modelling

**In the field of turbomachinery construction, CFD has, within the space of just a few years, become an indispensable tool for the optimization and new design of turbomachines, as well as for the elimination of fluid-mechanic problems in installed systems. The following paper demonstrates the basic knowledge as well as applied examples of the CFD simulations of process fans.**

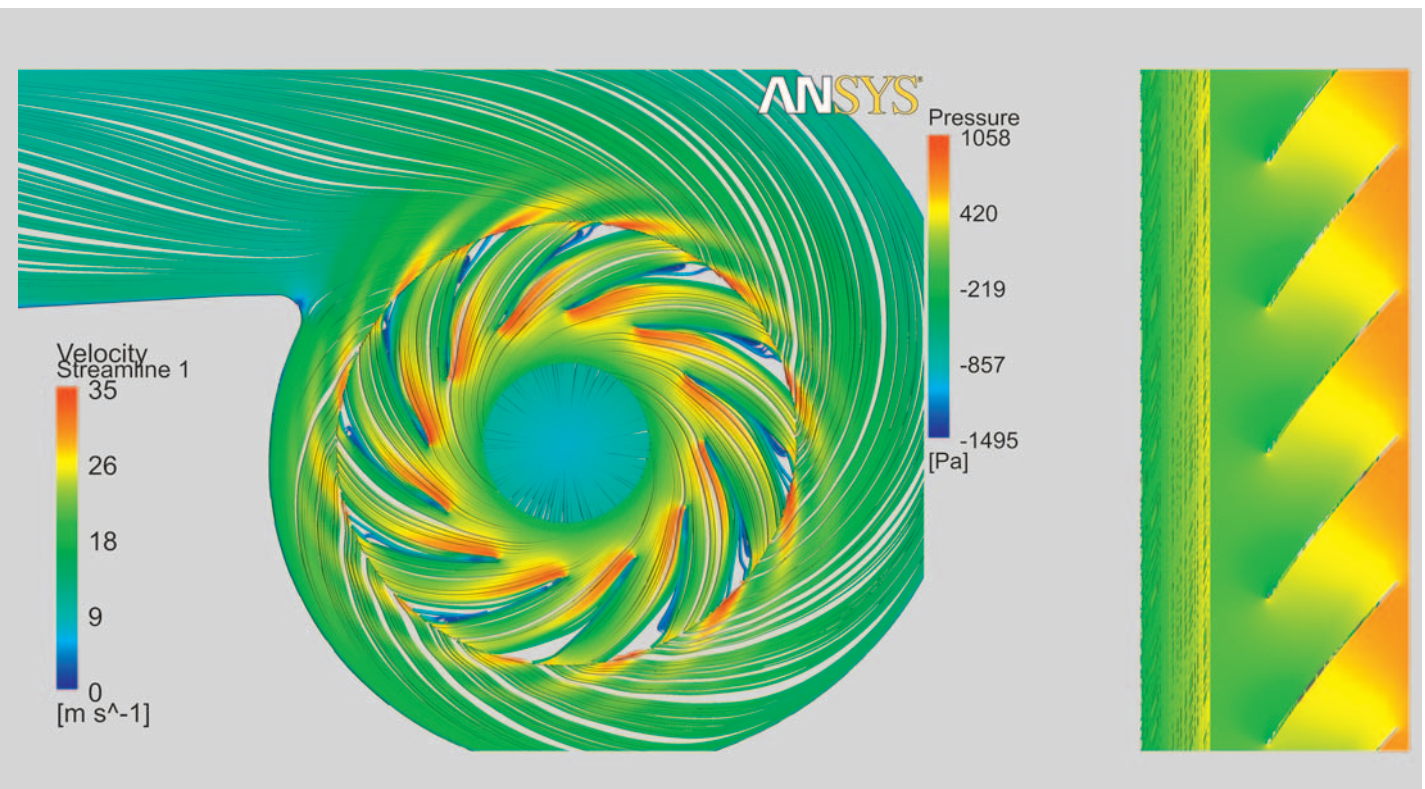
## 1 Introduction

Computational Fluid Dynamics (CFD) is a method that is used for the computerized calculation of technical flows with state-of-the-art accuracy. With this method, it is possible to make far more precise statements regarding the behaviour of a technical or natural flow than could be made on the basis of classical partly-empirical, test-based approaches. CFD has been employed on an economic scale since around 1973, when it was first applied in the aircraft construction sector. The underlying partial differential equations according to Navier-Stokes, which precisely describe the flow characteristics of a Newton fluid, have been known since the first half of the 19th century. The calculation work required for solving these differential equations is very complex: the flow volume has to be subdivided into a number of smaller par-

tial volumes that depend on the occurring flow gradients. These partial volumes then form the basis for the equations. For complicated geometries, such as fans, such calculations are therefore only feasible if they are performed by computer. This particularly applies to simulation of transient flow conditions, i.e. conditions that vary over time, as a number of related time steps have to be computed in order to obtain sensible results.

## 2 The mathematics behind CFD

The set of equations describing the processes of momentum, energy and mass transfer is known as the Navier-Stokes equations. These partial differential equations, which were derived in the early 19th century and have no known general analytical solution, can be solved numerically. For the numerical solution of this system of equations, the differential transport equations are first transformed into algebraic equations by so-called discretization. In the case described here, the Finite Volume Method is used for this purpose. This method is generally applied as the basis for simulation programs in the Computational Fluid Dynamics sector. In the FM method, a three-dimensional computational mesh is laid over the flow volume to be investigated. Determination of the distribution of a flow value  $\Phi$  thus takes place via discrete calculation points



(nodes). In this way, the continuous distribution of  $\Phi$  is represented by the  $\Phi$  values at discrete points. This creates an algebraic equation (finite difference equation) for every calculation point representing a single computation volume (cell). This algebraic equation is then used for determining the flow values. The differential equations from all the calculation points form a system of linked algebraic equations. This system of equations has to be solved with a numeric algorithm. A number of direct or iterative calculation methods are available for the solution of algebraic equation systems.

As the processing power of available computer technology increases, the application of CFD is becoming more and more interesting for even medium-sized and small companies, in spite of the high purchase and maintenance costs of the software and the present lack of specialist users. One of the biggest ad-

vantages of CFD vis à vis classical test bench methods is the ability to examine in detail any desired area within the simulated boundaries and thus to clearly identify and subsequently optimize the zones of rough flow. The field of application of the software thus stretches from the development of new products through the optimization of existing machine series and up to the elimination of fluid-mechanic problems in installed systems. The CFD user gains an understanding of flow phenomena that he can hardly hope to achieve through practical trials, or that can only be obtained at the highest academic level where the limiting factors are then the cost and time involved as well as know-how transfer.

At Venti Oelde the method is not only used for fan construction, but also for system engineering, for instance in the optimization of duct routes, stacks, filter plants,

cyclones or the air handling systems of recycling plants. In addition to such tasks as aerodynamic optimization, other fields of application for the software used at Venti Oelde are the simulation of mixing processes in stacks and heat transfers during drying processes and to determine the forces imposed on different structural geometries by the flow passing through them. These can then be employed as boundary conditions for structure-mechanical simulations in order to determine the maximum occurring stresses or for modal analysis to determine possible resonance effects caused by transient forces. In contrast to classical trials, a simulation entails no restrictions with regard to construction size, power consumption, possibly harmful conveying media or high temperatures. To summarize, it can be stated that given the current high technological level of turbomachinery construction, it is practically impossible

to make further improvements without the aid of numeric methods, because advancement of machine designs is generally accompanied by an exponential rise in costs compared to benefit.

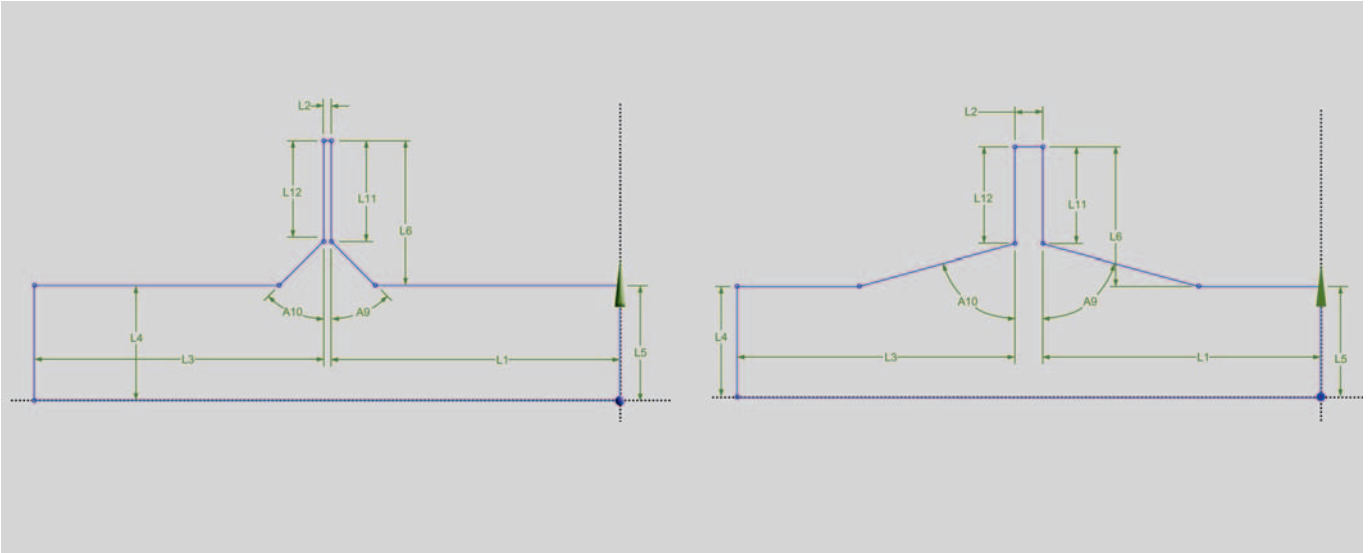


Figure 1 Parameterized layout with modification

### 3 Performance of a CFD simulation

To allow performance of a CFD simulation a number of work steps have to be carried out in the sequence described below. It is presumed for this purpose that the geometry data and the boundary conditions are known:

#### a) Creation

The first step is to create a three-dimensional model of the geometry using CAD software with 3D capability. It must be remembered at this stage that in the case of flow simulations the 3D geometry of the flow zone is needed and not that of the structure, as is usual in other types of analysis. Increasingly, the created 3D models are completely parameterized (Figs. 1 and 2) in order to ensure that

any modifications in the geometry can be quickly incorporated into the model by simply entering the new dimension, whereupon the entire model and any adjacent zones adapt themselves to the new dimension.

This brings a significant time saving when optimizing the model compared to the otherwise necessary new creation of a different model. In addition,

one or more two-dimensional sketches are produced. These can be used for creating three-dimensional models, for example by extrusion, rotation or by means of Boolean operations like cutting or merging.

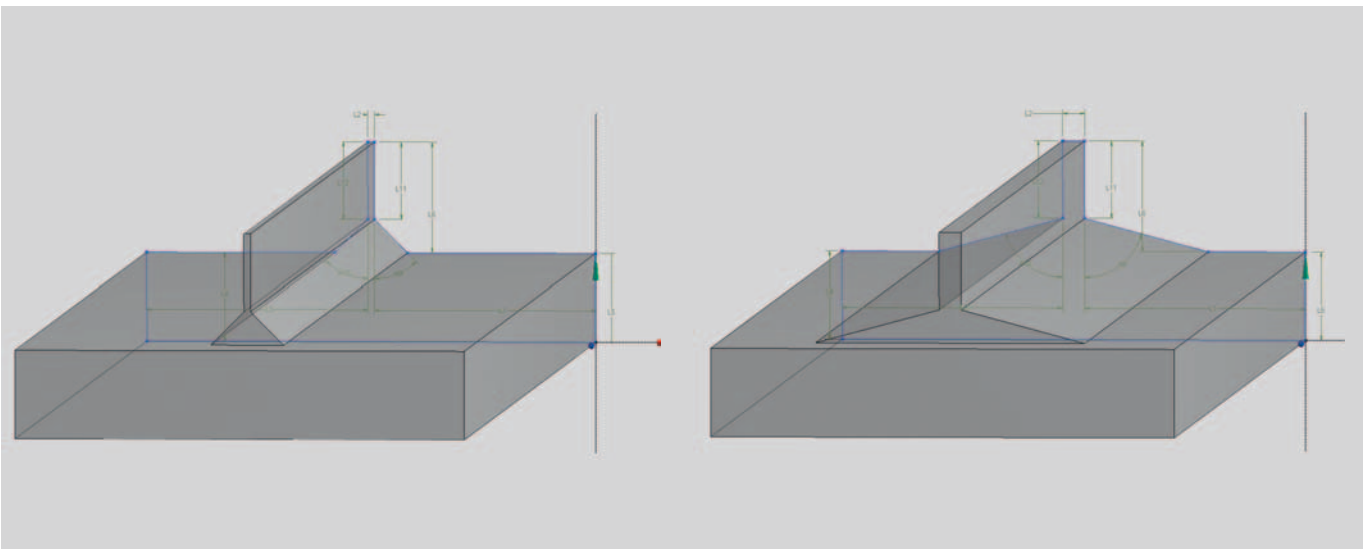


Figure 2 Adapted 3D models

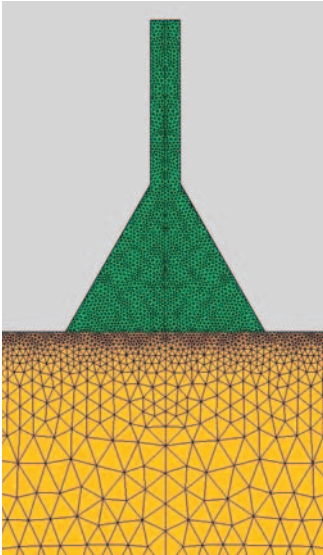


Figure 3 Unstructured computational mesh

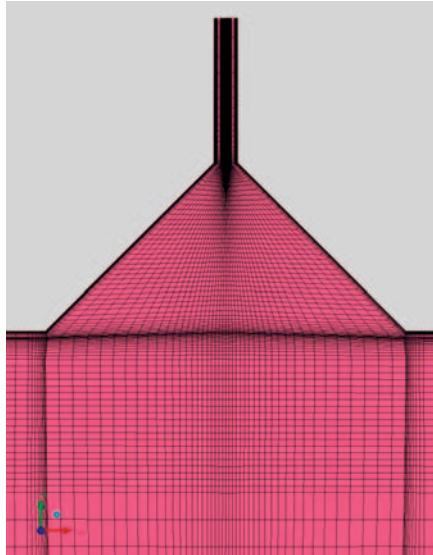


Figure 4 Block-structured hexahedral mesh

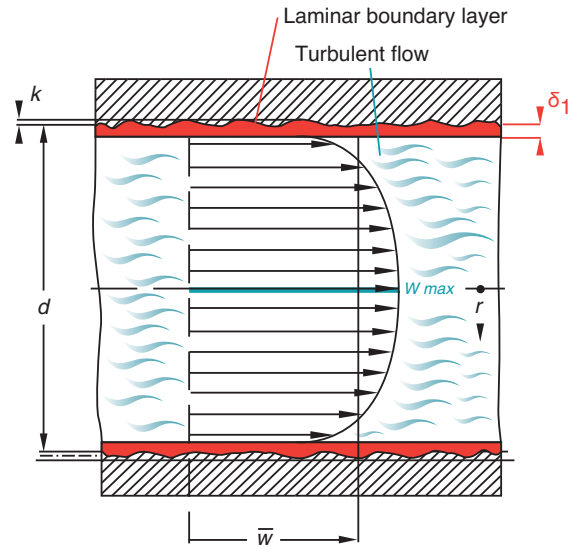


Figure 5 Flow profile

## b) Discretization

After the required 3D models have been designed, a three-dimensional computational mesh has to be created. The elements of this mesh form the basis for the control volumes that are needed, as already mentioned, for the partial differential equations, so that these can be iteratively solved in matrix form in accordance with the number of elements. Depending on the employed software, the user is generally able to create either a simple-to-generate, unstructured computational mesh (Fig. 3) or a considerably more difficult-to-generate, block-structured mesh (Fig. 4). As a rule, a block-structured mesh with a comparable structure resolution produces better results in significantly shorter processing time.

by the boundary layer, as the flow in this zone has to overcome not only the pressure increase but also the wall shear rate. If there is turbulent flow (and nearly all technical flows are turbulent) the boundary layer is only a few tenths of a millimetre thick and has a parabolic curve with respect to the velocity distribution profile (Fig. 5). The computational mesh for this zone must be given such a fine resolution that this parabolic curve is represented with sufficient accuracy. In a normal view of the wall, this requires between 8 and 14 elements in the boundary layer zone (Fig. 6).

For this reason CFD studies at Venti Oelde are, with very few exceptions made for time reasons, all based on block-structured computational meshes. Care has to be taken that the flow dynamic boundary layer is given an adequate resolution, because – for instance – flow disruptions are always caused

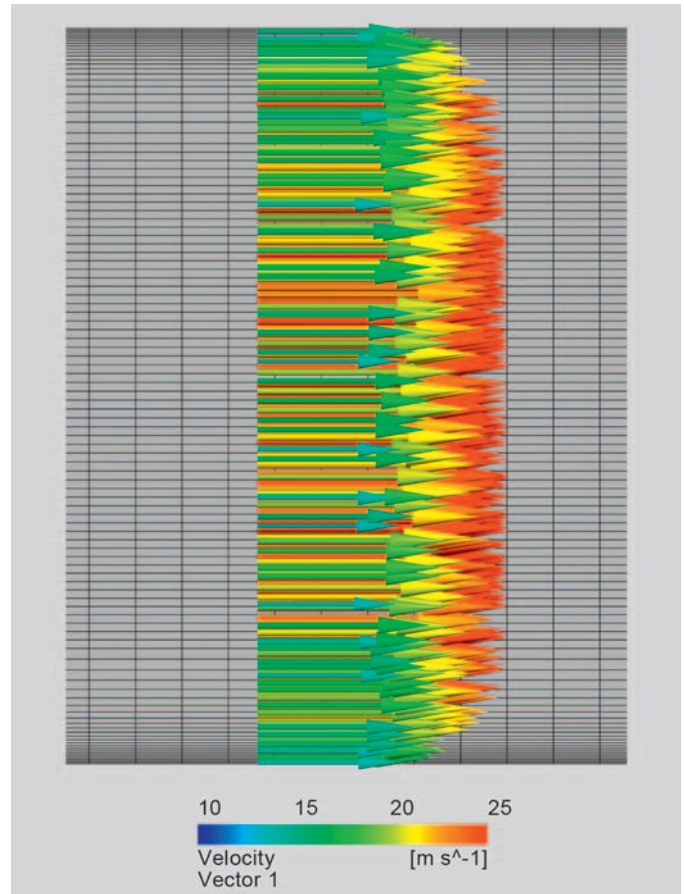


Figure 6 Boundary layer discretization with vector arrows

### c) Simulation

After the 3D model and subsequently the computational mesh have been created, the simulation software (ANSYS CFX) is brought into use.

ANSYS CFX consists of three modules: ANSYS CFX-Pre (Fig. 7) for definition of the boundary conditions of the simulation, ANSYS CFX Solver Manager (Fig. 8) for iteratively solving the partial differential equations until a predefined convergence criterion is reached, and ANSYS CFX Post (Fig. 9) for visual and numeric evaluation of the simulation results.

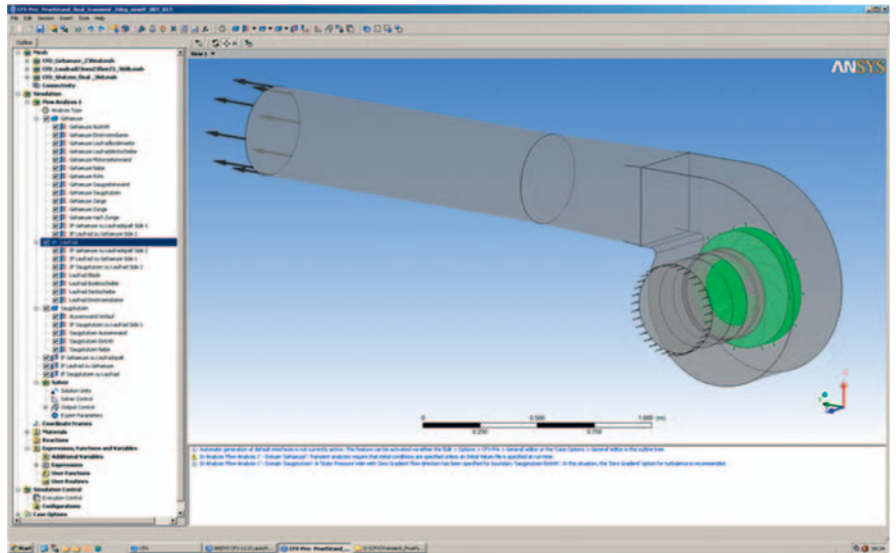


Figure 7 ANSYS CFX-Pre



Figure 8 ANSYS CFX Solver

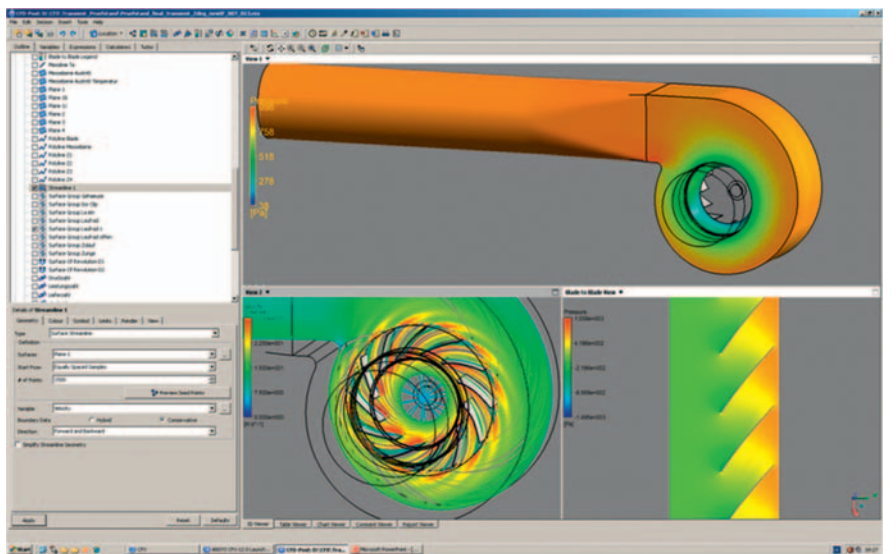


Figure 9 ANSYS CFX Post

## 4 Applications

### 4.1 Optimization of the intake flow of a stack

In this case, after modification of a pressure-side duct leading to a stack and containing several branches (Fig. 10), vibrations occurred on one of the fans, causing an automatic safety shutdown. These vibrations had not been experienced prior to the modification. As it could be presumed that the vibration problems were a consequence of non-optimum design of the supply duct, this was analysed in detail by CFD. The stack is around 70 m high while the horizontal duct is approx. 3 m high, 2 m wide and has a total length of approx. 25 m. The mentioned vibration problems affected the fan upstream of the marked duct. It was established that the fan vibrations were caused by poorly designed cross-sections. The measured flow velocity in the horizontal section of the duct was 22 m/s, while that in the vertical section marked in Fig. 10 was only approx. 2-3 m/s. With approximately comparable duct cross-sections, the flow momentum (mass flow  $\times$  velocity) of the flow compon-

ent coming from the left was considerably higher than that entering the collecting duct from below. As can be clearly seen from Fig. 11, this causes a severe constriction of the air stream entering the collecting duct from the bottom left. As a consequence, there were pressure fluctuations in excess of 250 Pa in this supply duct (Fig. 12), and these throttled the fan so severely that it reacted with significantly increased vibration velocities.

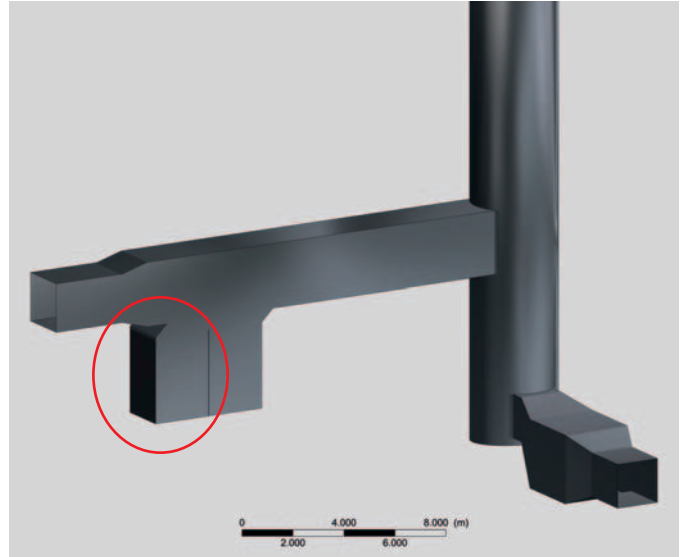


Figure 10 Stack geometry with marked problem zone

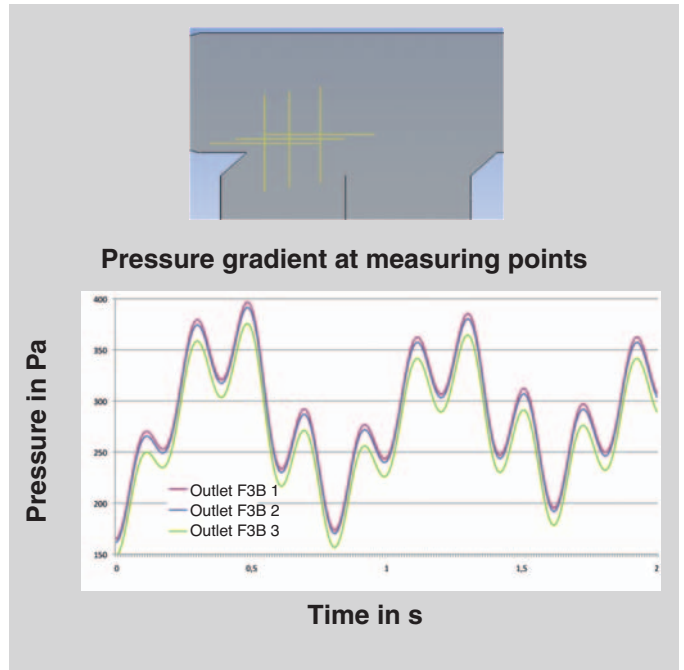


Figure 12 Pressure pattern at measuring points

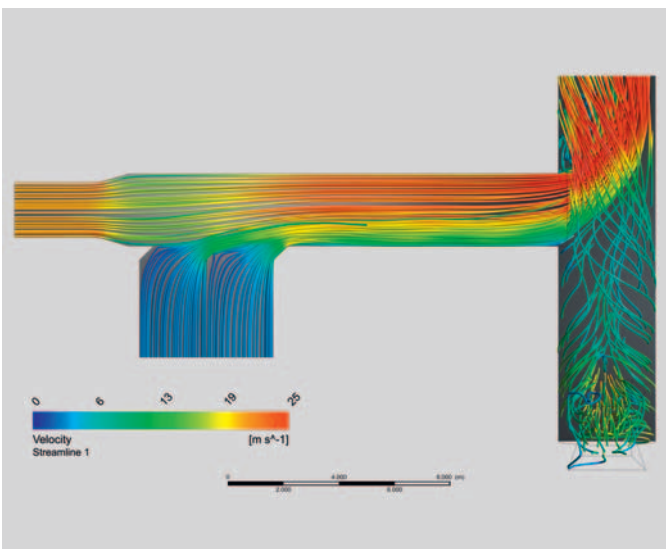


Figure 11 Velocity distribution in stack

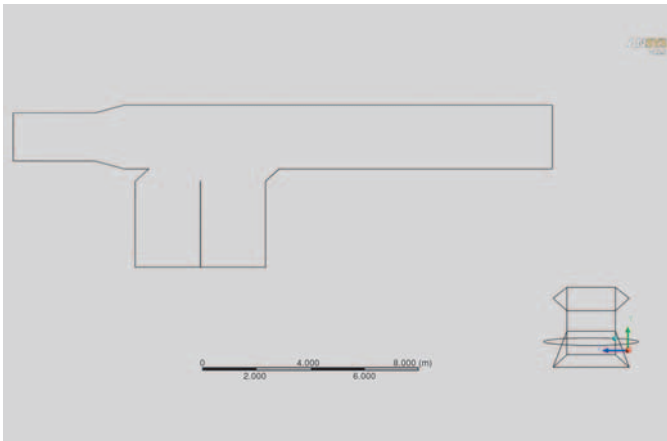


Figure 13 Original geometry

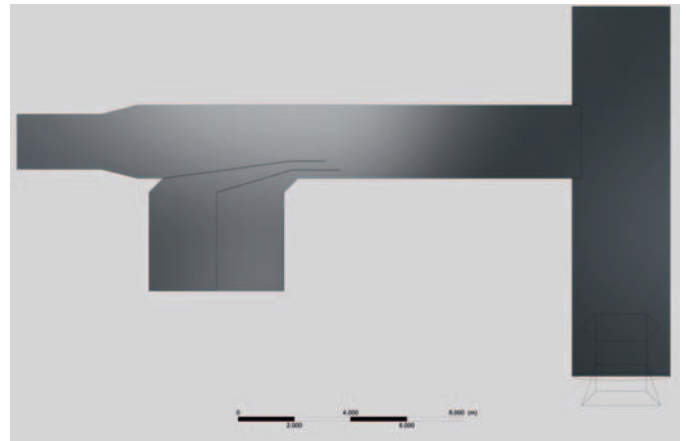


Figure 14 Duct with baffle plates

As the described unsatisfactory conditions had to be remedied without significantly interrupting operation or altering the design, it was necessary to eliminate the problem by the simplest means possible. Without altering the duct geometry (Fig. 13), this was achieved by installing two baffle plates (Fig. 14), which accelerated the flow from the two lower supply ducts to the extent that the velocities of all three gas streams are identical at the point where they meet. This measure succeeded in optimizing the velocity distribution (Fig. 15) compared to the

original situation (Fig. 11), so that the gas streams meet with identical velocities and no longer negatively affect each other. Also, the pressure fluctuations measured at the points shown in Figures 21 and 25 were reduced from 250 Pa to below 10 Pa. Subsequent to the conversion work, the upstream fan could be run up to its rated speed without any problem. The vibration characteristics were so greatly improved by the conversion work that measurements taken at the fan detected no further effects on fan operation.

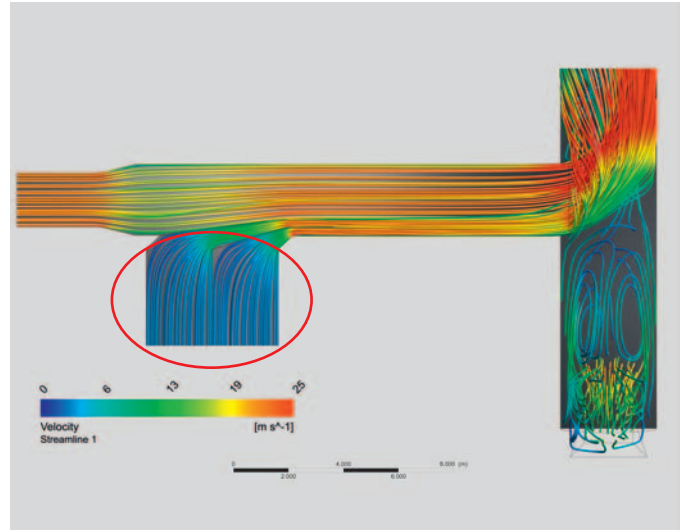


Figure 15 Velocity with optimized geometry

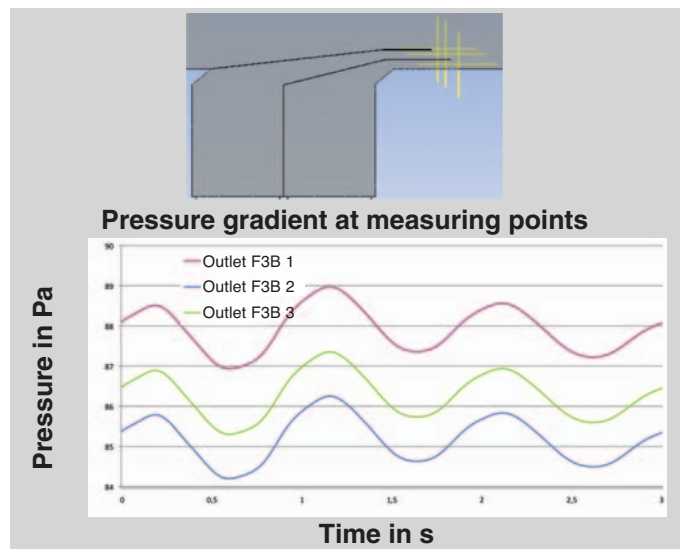


Figure 16 Pressure gradient with baffle plates

## 4.2 Optimization of an induced draught fan

In addition to optimization of the fan itself, the upstream and downstream flow situations are of great importance for the troublefree operation of the fan and achievement of the maximum possible efficiency. If, for instance, the incoming flow is turbulent or swirling because of poorly designed bends or changes in cross-section, this inevitably results in a worsening of the fan's operating characteristics. Disturbances in the inflow and outflow zones have particularly serious effects in the case of fans with especially high efficiency levels > 80 %, as such fans depend on a non-swirling inflow in order to achieve their efficiency figures. As well as aiming at optimizing the fan impeller and casing it therefore makes absolute sense to analyse the inflow and outflow situations. This application is therefore concerned with the optimization of the induced draught flow of a double-inlet fan. Figure 17 shows the system arrangement drawing. The 3D model derived from this drawing is shown in Figure 18. The very conspicuous sharp-edged transitions are very detrimental to the achievement of smooth flow guidance. This becomes plain in the subsequent evaluation of the flow simulation (Fig. 19), which reveals extensive flow disruption in the rear area that is also unstable, i.e. it oscillates between the left-hand and right-hand sidewalls and thereby obstructs a portion of the branching-off rear induced draught flow. The pressure drop of this system is correspondingly high, as is illustrated by the total pressure profile shown in Figure 20. From the inlet until the end of

the first branch the pressure drop is approx. 260 Pa. From the inlet until the rear branch the drop even reaches 430 Pa because of the flow disruption at the rear of the horizontal duct, as shown in Figure 19.

In Figure 20 the points of particularly high pressure drop are highlighted. The black circles indicate unfavourable situations because this design involves transitions that are too sharp-edged and thus prevent the flow from following the contours, resulting in turbulence. This causes pressure drops and thus leads to additional expenses for electrical energy. By contrast, the white circle indicates a duct end producing an inefficient fluid flow. At the marked rear end of the duct the flow forms a vortex (Fig. 19), which has several negative consequences. Firstly, the flow is subjected to pressure fluctuations because the vortex, as already described, wanders between the left-hand and right-hand duct walls. Secondly, a portion of the vortex is sucked through into the fan (Fig. 21, view from below into the duct), so that the fan has to cope with strongly swirling inflow air and therefore loses efficiency and may suffer from mechanical vibrations (resonance).

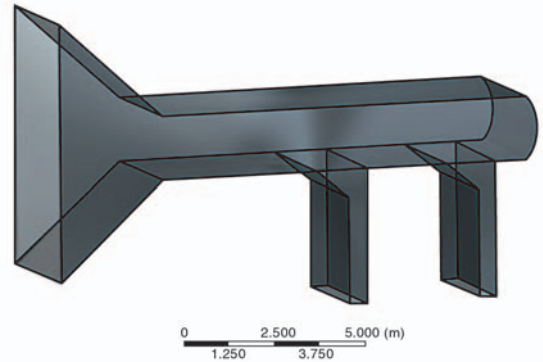


Figure 18 3D model of induced draught

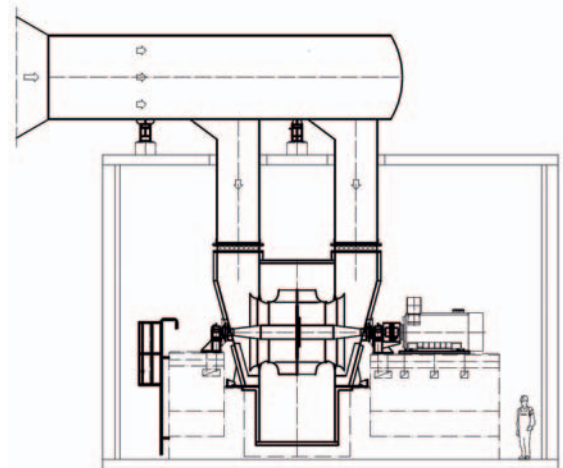


Figure 17 System arrangement drawing with induced draught fan

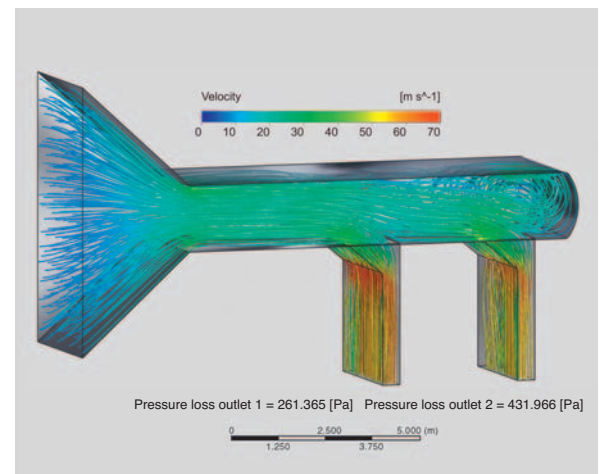


Figure 19 Flow lines

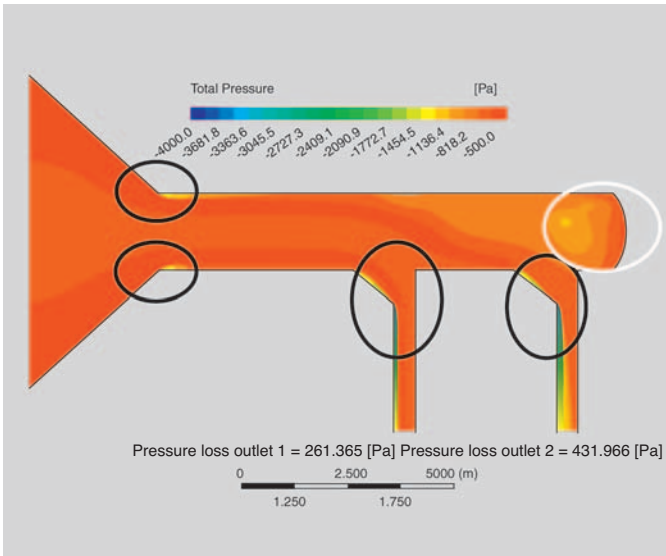


Figure 20 Total pressure profile with original design

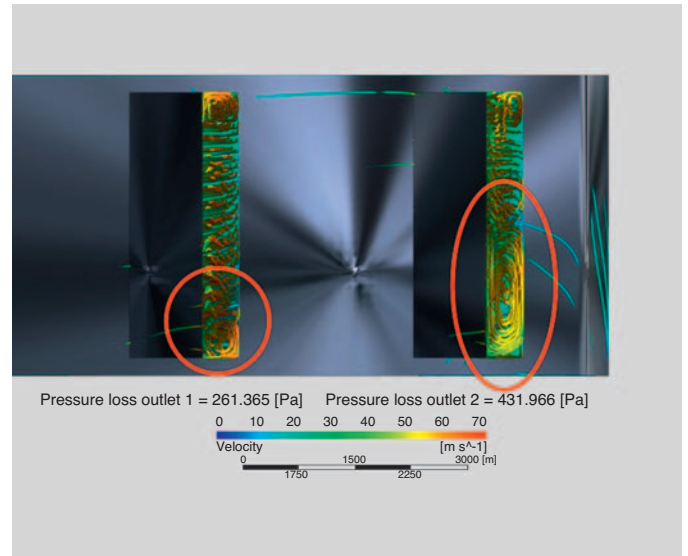


Figure 21 Strong swirl with original design

Optimization of the geometry involved redesigning the critical points highlighted in Fig. 20 so that the cross-section transitions were smooth. This redesigning succeeded in reducing the pressure drop at the front inflow duct to the fan by a factor of four, from 261 Pa to 66 Pa (Fig. 22), while the pressure drop at the rear inflow duct that had been caused by the poorly-designed horizontal duct end (Fig. 20, white circle) was even reduced to little more than one sixth the original figure. The severe

swirling of the fan intake air (Fig. 21, red circles) was also significantly reduced (Fig. 23), enabling the fan to achieve its rated performance figures.

When the flow lines of the original duct design and the optimized duct design are compared (Figs. 24 and 25) it is obvious that the flow turbulence has been completely eliminated and the flow velocity of max. 70 m/s of the original design has been reduced to approx. 55 m/s, resulting in decreased swirling and also

in a lower pressure drop. The mean pressure drop saving of 275 Pa at a volume flow of 500000 m<sup>3</sup>/h significantly cuts the power consumption. The resultant power saving of 49 kW, assuming 24-hour operation at an electricity price of 10 cent per kWh, leads to an annual electricity cost saving of 43000 €.

Related purely to the flow through the duct, the pressure drop coefficient  $\zeta$ , which is decisive for the system pressure drop, has been halved.

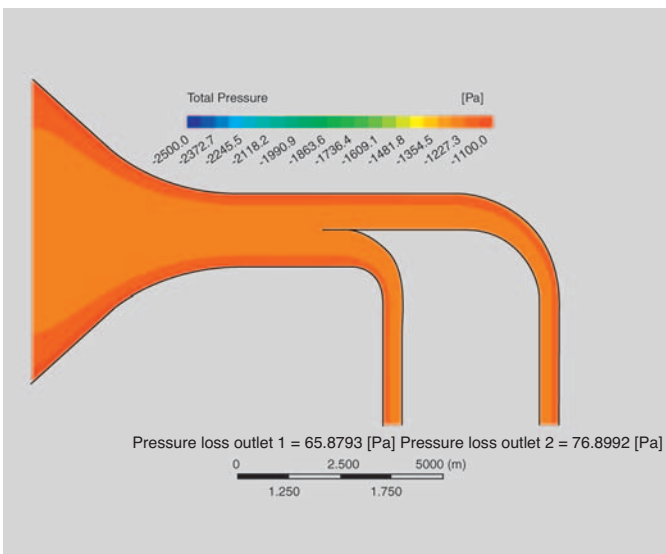


Figure 22 Total pressure profile after optimizing design

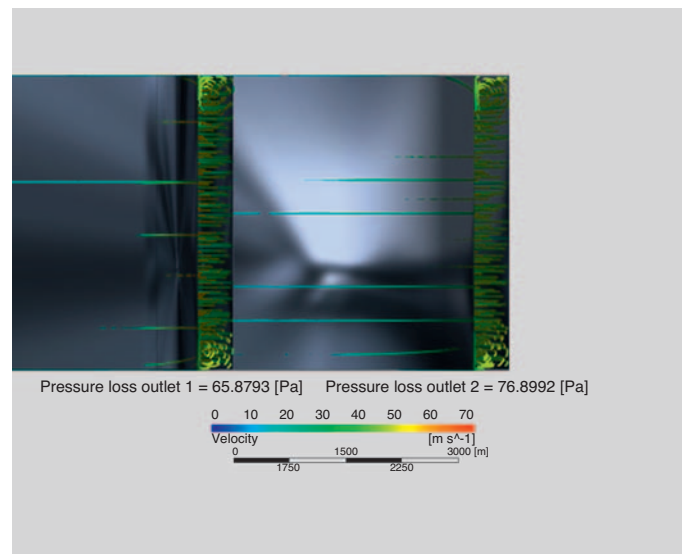


Figure 23 Swirl after optimizing design

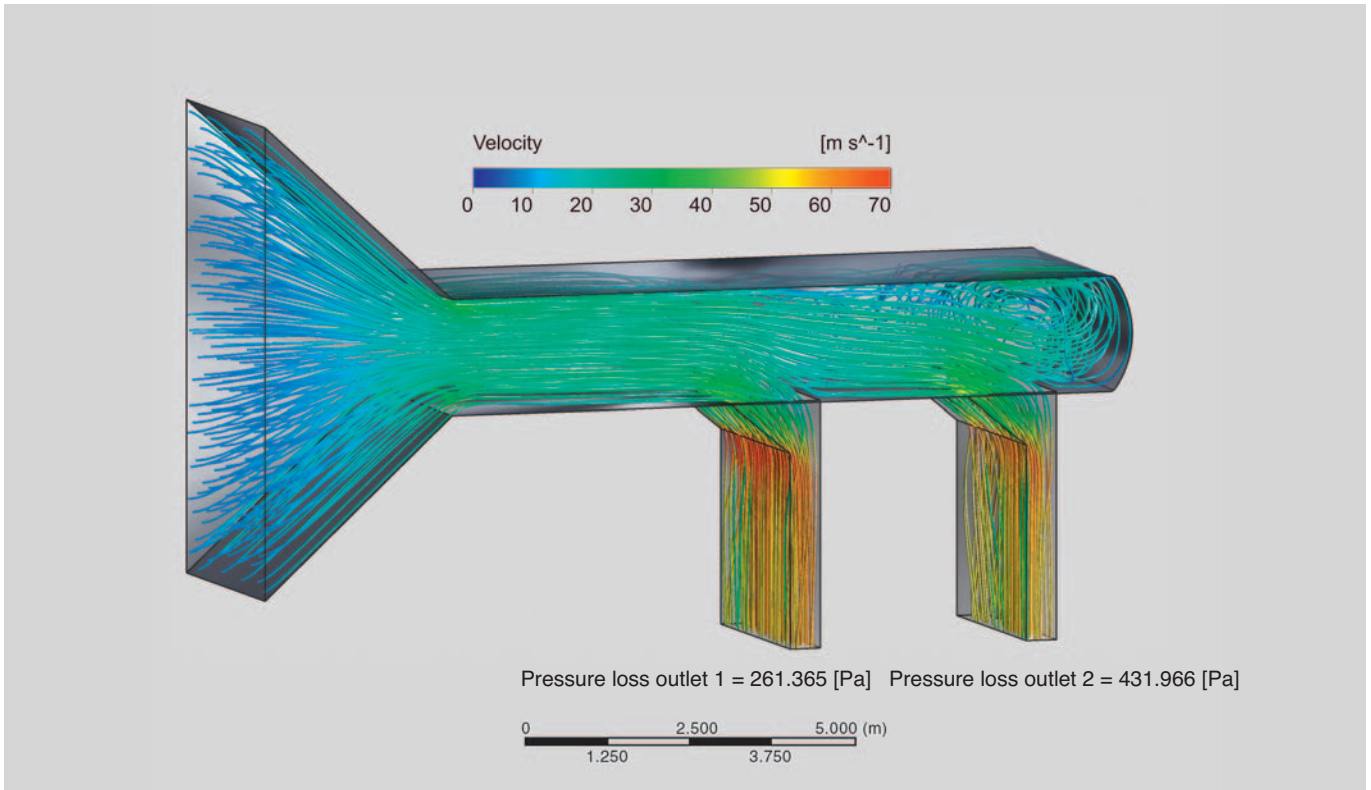


Figure 24 Flow lines of original design

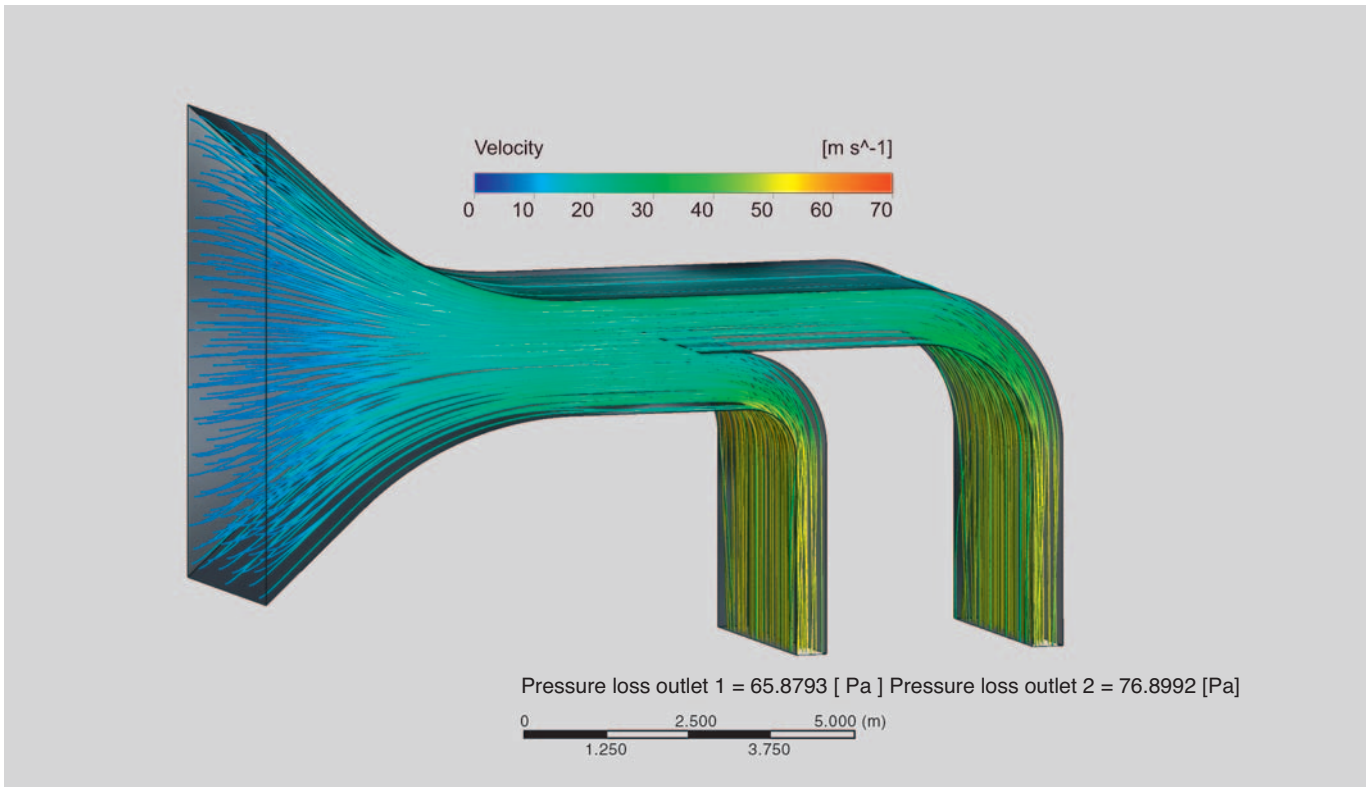


Figure 25 Flow lines of optimized design

### 4.3 Optimization of a large double-inlet fan DHRV 50

During the introduction phase of the software, its computation accuracy was tested by comparing simulation and measurement using a test-bench fan of type HRV 63S and it was found that the simulation deviated by less than 1% from the measured values.

The first results of an ongoing optimisation of a large double-inlet fan type DHRV 50B-2000 are presented in the following. This is an ongoing project which has the aim of determining the optimization potential of the DHRV 50, particularly at high peripheral speeds > 150 m/s. The background to this is the fact that for some years now, clients have been requesting higher and higher pressure differences together with a high volume flow. These requirements can only be fulfilled if the speed and thus the peripheral speed of the fan impeller are increased. This means that the speeds and pressures occurring in the fan reach and exceed the design limits normally defined in fan specifications of Mach 0.3 or 100 m/s at a pressure ratio of

max. 1.3. If these limits are exceeded, which takes place more and more frequently, it is no longer possible to achieve maximum efficiency ratings by means of the classical empirical design methods for fans (acc. to Prof. L. Bommers or Dr.-Ing. Bruno Eck). However, due to the substantial increase in energy costs and also because of the energy efficiency directives aiming at a reduction in CO<sub>2</sub> emissions, it is imperative to achieve the highest possible fan efficiency rates. To enable this, it is either necessary to plan and carry out extremely expensive, time-consuming and labour-intensive test series or to employ CFD tools, which are the state of the art in turbomachinery construction. One great advantage when using CFD for designing and optimizing fans is that it provides a detailed analysis of every flow zone. Many of the flow zones that are of interest to the design engineer, for instance the leading edges of impeller blades, either cannot be represented in sufficient detail by conventional measuring techniques or would demand a disproportionate amount of

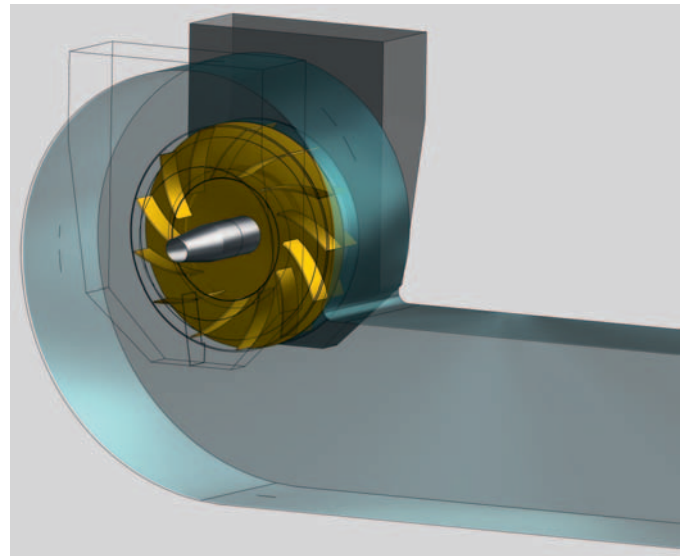


Figure 26 Geometry DHRV 50B-2000

work and expenditure. For example, measuring devices fitted onto the rotating impeller would influence the flow in a way that would falsify the measurement results or are so complex that they can only be applied in academic studies (light sectioning principle). However, in order to improve the efficiency rates beyond the current status of > 80 %, it is essential to obtain a well-founded knowledge of flow dynamic processes in the impeller and fan casing. Due to the ability of CFD to numeric-

ally describe the behaviour of a Newton fluid in a physically correct manner, CFD is currently the most effective tool for achieving further optimization of turbomachines. Particularly in the field of compressor construction, which is closely related to that of fan construction, CFD has become an indispensable tool for analysing, for example, impellers with twisted blades, twisted leading edges and raked trailing edges. The project described below was the first step in the optimization of a

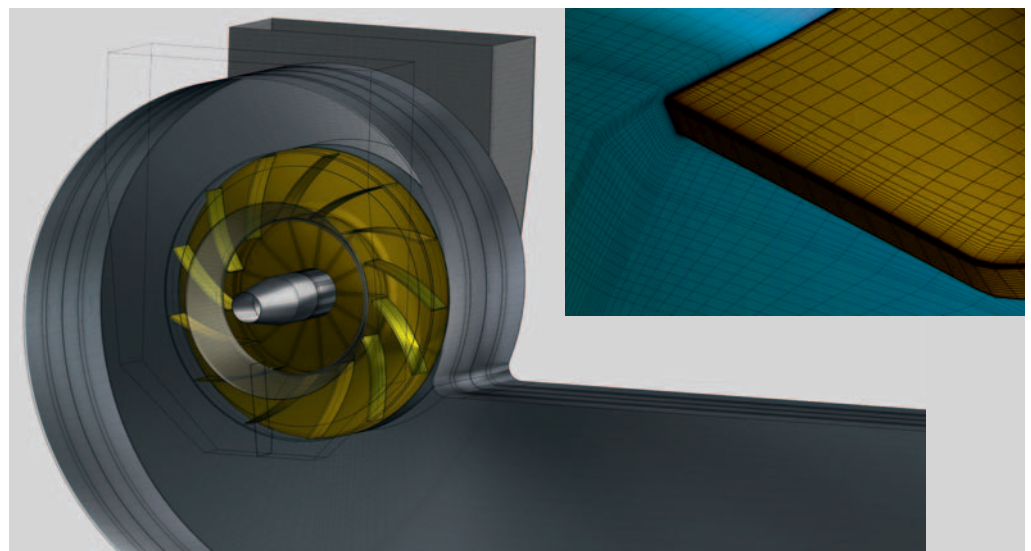


Figure 27 Mesh of DHRV 50B-2000 with detail of leading edges of impeller blades

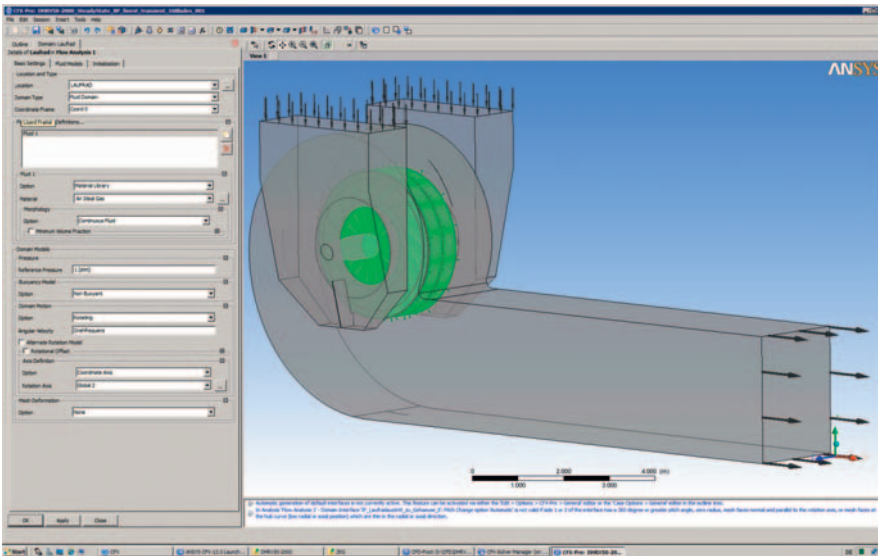


Figure 28 Specification of the boundary conditions

large double-inlet fan of type DHRV 50B-2000.

First of all, a 3D model of the inlet box, the impeller and the volute casing (Fig. 26) has to be created and then provided with a computational mesh. During the discretization process, special attention is paid to the most uniform possible distribution of the mesh elements, which has to be oriented to the occurring flow gradients. For instance, the computational mesh has to assure great detail on all the walls (detail in

Fig. 27), in order to represent the earlier described boundary layer with adequate precision. In the case of the described large double-flow fan of type DHRV 50B-2000, which has an impeller with an outer diameter of 3.2 m, the total number of computational mesh cells in all zones is approx. 8 million elements. If, as in the described case, there are adjacent stationary and rotating zones, these have to be separately represented in the model and separately discretized. This is because the computa-

tional mesh of the impeller rotates during a transient simulation through a defined angle for each of the individual computing steps (usually by 1-5° depending on the number of blades), while the inlet boxes and the volute casing remain stationary. When the boundary conditions are being entered, these zones then have to be connected by interface (Transient Rotor Stator Interface). After the boundary conditions have been specified (Fig. 28), including a rotational fre-

quency of 893 min<sup>-1</sup>, a pressure difference of 8500 Pascal and a mass flow of 145.13 kg s<sup>-1</sup> at a process temperature of 300 °C, the transient calculation of the case can be started (Fig. 29).

After the specified convergence criterion has been attained and the simulation has been concluded, the results are evaluated. Figure 30 depicts the flow lines in the impeller and the casing. Every flow line represents the track of a single fluid particle through the entire calculation zone. The colour of the flow line indicates the local velocity of the fluid particle. With the aid of this flow line diagram the design engineer can identify points with particularly high flow disturbances, excessively high velocities or excessively sharp deflections. The diagram is also used for checking whether, for example, the air flows to the impeller blades or



Figure 29 Solution of the differential equations

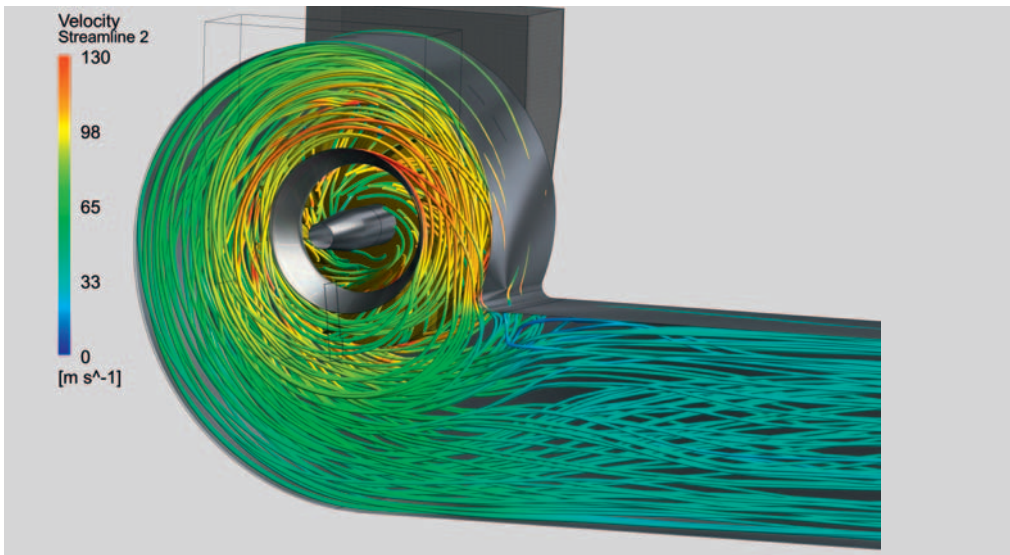


Figure 30 Flow line diagram

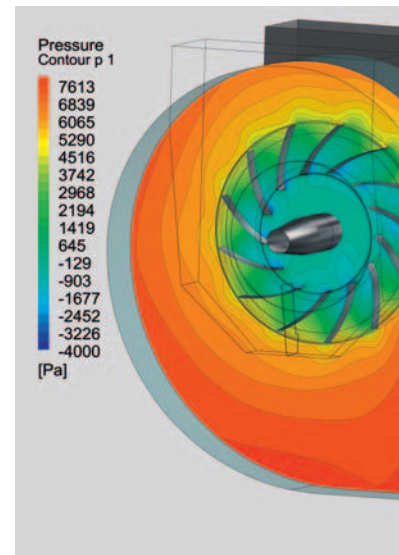


Figure 33 Pressure distribution in impeller and casing

the casing cut-off (the point at which the flow is divided) as planned.

In addition to the flow lines, a diagram of the flow vectors is a suitable way to visualize the characteristics of a technical flow. The flow vectors indicate both the magnitude of the velocity and its direction. This is of great importance, especially in the case of flow turbulences. Figure 31 shows the vectors at the leading edge of the impeller. Figure 32 shows the flow vectors at one of the most critical fluidic zones of

the casing, the casing cut-off, where the flow is divided. It is extremely important that the flow divides at the centre of the cut-off, as shown in Fig. 32. If this does not happen, the result is loss of fan efficiency, vibrations and a significant increase in sound pressure level.

Figure 33 shows the pressure distribution in the impeller and casing. The right-hand image shows the inlet box. The pressure distribution reveals that approx. two thirds of the pressure increase take place in

the impeller while approx. one third takes place in the casing, which thus acts as a collecting plenum and diffuser. One of the aims of the ongoing optimisation of fan type DHRV 50B is to achieve complete uniformity of the currently not entirely homogeneous pressure distribution at the periphery of the casing.

The optimization potential still existing in the casing can also be seen from the flow lines shown in Figure 34. To enable easier interpretation of the flow lines than is possible on the

basis of Figure 30, these were projected as 2D flow lines onto a plane inserted in the centre of the casing. This clearly shows that the velocities in the area of the cut-off (red circle in Fig. 34) are slightly too high and that the fluid does not flow away in fully logarithmic manner (black circle). In order to determine the optimization potential at these points, different designs are currently being simulated and compared. For instance, the casing width is being varied in a number of increments in order

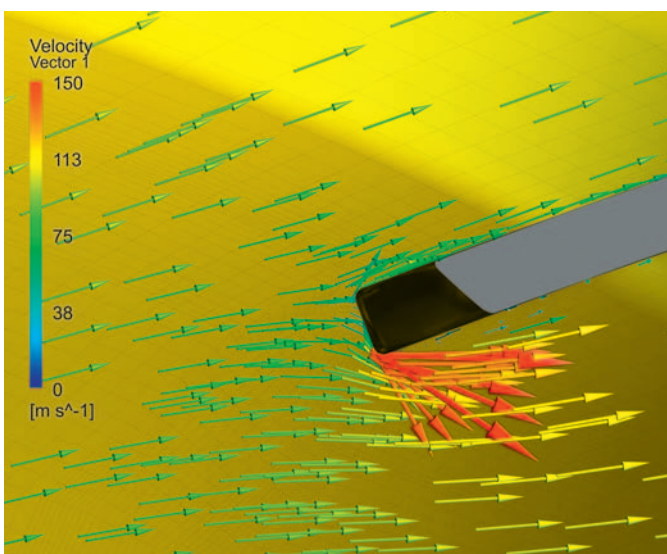


Figure 31 Vectors at leading edge of impeller

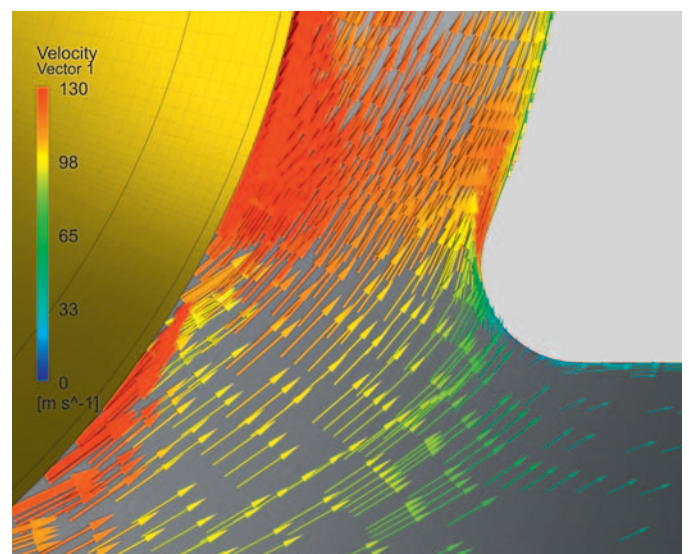


Figure 32 Vector arrows on casing cut-off

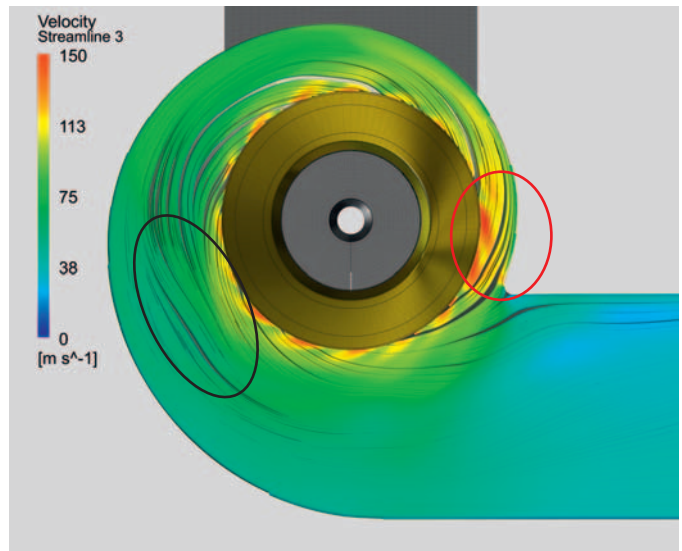
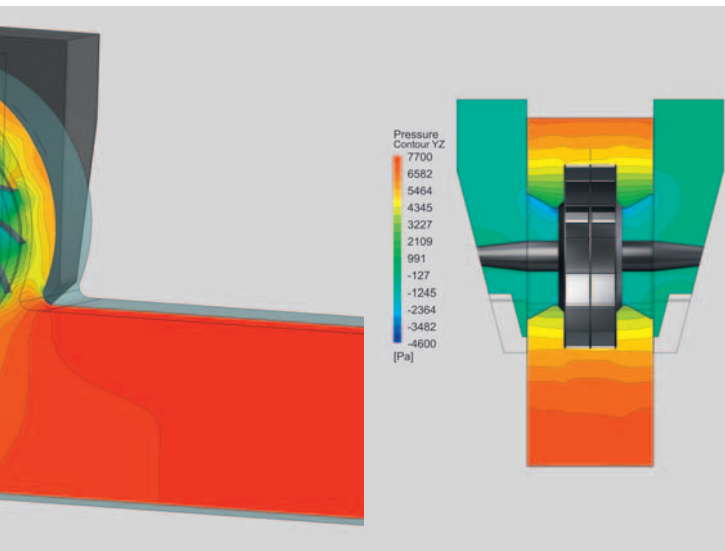


Figure 34 Flow lines in 2D

to improve the flow and to achieve a saving in material and thus in machine weight. After this test series has been concluded, the logarithmic function used for designing the casing periphery will be subjected to a series of checks in order to achieve improved flow guidance. In addition to the casing, which offers the greatest optimization potential, the impeller is also being analysed. One possibility that has only recently been added to the software is to represent variables as volumes in specified ranges. Using this software capability, Figure 35 was generated. This depicts the total pressure in the impeller within a range of -4000 to -6000 Pascal. The shape assumed by the individual volumes corresponds very closely to photos of material incrustations on impellers conveying dust-laden media (Fig. 36). If these areas

are selectively optimized in order to achieve a more uniform total pressure distribution, the amount of incrustation will certainly be reduced.

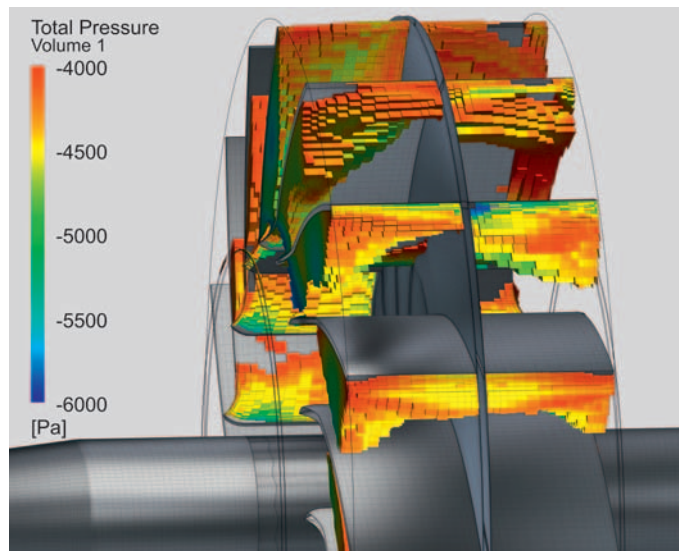


Figure 35 Total pressure volumes



Figure 36 Raw meal caking

## 5 Conclusions and prospects

Within a period of one year after the test phase, the CFD method had completely established itself at Venti Oelde. In addition to its primary function as a tool for optimizing and advancing the design of fans and their inflow and outflow zones, the software plays an

important role in system engineering and construction because, besides enabling purely aerodynamic assessment of a flow, it also allows calculation of mixing procedures or heat transfers and distributions.

In the field of turbomachinery construction, CFD has, within the space of just a few years, become an indispensable tool for the optimization and new design of turbomachines, as well as for the elimination of fluid-mechanic problems in installed systems.



Ventilatorenfabrik Oelde GmbH  
P.O. Box 37 09  
D-59286 Oelde  
Phone: +49 25 22 75 - 0  
Fax: +49 25 22 75 - 2 50  
info@venti-oelde.de  
www.venti-oelde.de

- ▶ Industrial fans
- ▶ Dust collection and process air cleaning plants
- ▶ Exhaust air treatment plants
- ▶ Ventilating, heating and air conditioning plants
- ▶ Recycling and waste processing plants
- ▶ Surface technology