K. Kołodziejczyk, T. Zacharz
University of Mining and Metallurgy, Cracow, Poland

APPLICATION OF CFDESIGN SOFTWARE TO VISUALIZATION OF FLOW SUSPENSION IN A PROTOTYPE SETTLING TANK

Подано приклад використання CFD для моделювання характерного розподілу суспензії в прототипі приладу для системи зрошення полів, який використовується для відстоювання та групування суспензії. Прототип приладу містить два типи процесів – прямотечійний в зоні концентрації та протитечійний в зоні відстійника. Для потоків 80 м^3/год було проведено комп'ютерне моделювання. Використана програма CFdesign для моделювання розподілу швидкості в пристрої системи зрошення.

Authors present the example of exploitati on CFD to model a proper distribution of suspension in a flooding system of the prototype device, which is used for clarifying and concentrating the suspension. The prototypical device combines in itself two kinds of processes – co-current in the concentration zone and counter-current in the clarifying zone. Computer simulations were made for flows 80 m^3/h. CFdesign software was used for modelling of velocity distribution in the flooding system of the device.

Introduction

CFD – Computational Fluid Dynamics – allows for modelling and computing the flow parameters, which are very hard to get in the analytical way. The usage of CFD enables simulation and flow computation in the design phase. Owing to that it is not necessary to build and to make tests on real device. Moreover, the price of the project decreases when verification of the working device is done in a design stage. It also allows for the realisation of the project in shorter time.

The Department of Technological Equipment and Environmental Protection conducts the research at the new prototype device, which combines in itself two kinds of processes – co-current in the concentration zone and counter-current in the clarifying zone. The consequence of the research is the design of the prototype device whose work is based on the processes mentioned above. A very important design stage is the proper modelling of suspension flooding system of the device. This has a very big impact on afterwards device working and on effective usage of multi-flux lamella. In the considerate case, the flooding system consists of two diffusers symmetrically located. However, as the system was symmetrical, only one diffuser was taken to the simulation. The space between upper and lower layer of the multi-flux lamella was taken to the consideration during the initial simulation. To proper functioning of devices, the steady velocity distribution in the output of the diffuser is required. As a result, non-uniform velocity profile would cause a diversified loading of the multi-flux lamella and worst efficiency of the device. The CFD methods were used for modelling of velocity distribution in the flooding system of the device. Modelling and verification of a proper device work was done in a few stages. Firstly, authors of the article, so as to model the flow, designed a computer model of a settling tank in SolidWorks. The shape and size of the settling tank were made for flows 80 m^3/h. Secondly, the created geometry was exported to CFdesign where the proper boundary conditions were put on, and finally, the flow was modelled. Next, they performed simulation and they analysed velocity distribution for a system without and with different shapes and numbers of a flooding stators. Thanks to the graphic interface, visualisation of the movement stream path appeared to be possible. Having the precise image of the suspension flowing through the flooding system, it was possible to modify the construction (it means the change of shape and number of flooding fences) so as to get uniform velocity in the whole settling tank.

System geometry

As it was mentioned in the introduction, the system geometry was done in SolidWorks 2001 plus. The settling tank was designed for flows 80 m^3/h. At the Fig. 1 the designed settling tank is presented; the limiting outline of device is 3x3x4m. Consequently, the diffuser angel of flare in ranging from 8° to 14° cannot be used. Otherwise, the length of the diffuser would be longer than the settling tank overall dimension. In the prototypical device the 90° diffuser angel of flare was assumed. Another assumption was that the construction and size of diffuser could be modified in the boundary limits, which would cause better working conditions of the device. Fig. 2 presents the initial shape of the flooding system.

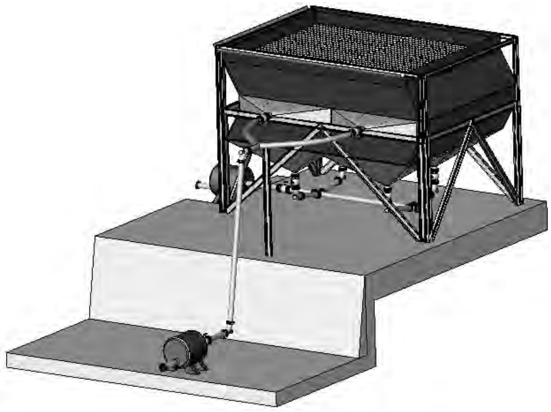


Fig. 1. Settling Tank

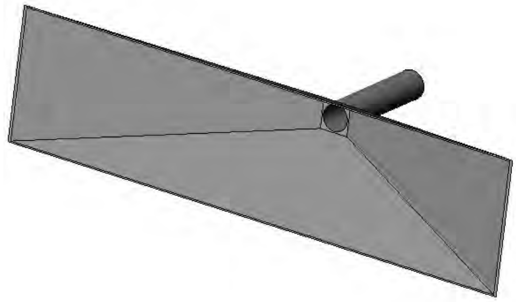


Fig. 2. Diffuser

Procedures

The first stage of simulation was the creation of geometry. The volume of diffuser, which is fluid, was our geometry. The thickness of diffuser was not taken under the consideration, because during the fluid flow through the diffuser the heat exchange with environment does not bigger influence on the velocity distribution. Therefore, it was the reason so as not to take into account the outline geometry and the change of temperature.

The next step was to import geometry to CFdesign Domain – program interface; a so-called preprocessor – in which boundary condition and mesh are created. The geometry export was done with the usage of a parasolid format. It is one out of two proper formats, which can be read by the above software. Next, the flow in CFdesign Domain was modelled – boundary conditions were put on and finite elements mesh was modelled and generated. The model was accepted and exported to Solver, where the type of flow, property of fluid and analytical model were defined. The analysis and solving were the final stage of modelling.

Then, the model was opened and analysed in CFdesign Display, a so-called postprocessor, where authors could present the precise image of the flow through the flooding system and various type of graphs. The data can be saved in ".csv" format.

Parameter of analysis

One of the analysis steps includes matching the fluid type, flow parameter and computational parameter that is, selecting a proper computational model. While carrying simulations, it is very important to try to produce a faithful copy of reality. However, in some cases, it is not possible to be faithful, either because of the software computational abilities, or the insufficient knowledge of the parameter impact to the modelling object. The faithful image of reality may cause that the computational time is too long and is not adequate to the quality of the received solutions. Therefore, so as to shorten the time of the analysis and to decrease the difficulty level, some parameters, having marginal influence on the being analysed flow, (e.g. simplification of symmetrical models) could be simplified.

Fluid Parameters

A Diffuser, for which the flow was modelled, is a part of the device to concentrate the suspension. It is the reason why the calculative medium should be the suspension with parameters corresponding to the brief foredesigns. The first approximation was the application of the fluid with the substitute parameters corresponding to the suspension parameters. The above simplification was possible because of two reasons: velocity in the diffuser was high and the stay time of suspension in the diffuser was very short. Thus, sedimentation should not take place. The aim of the simulation was also to get a steady velocity distribution on the output of the diffuser.

Computational viscosity of the fluid was labelled a substitution viscosity of suspension. It was calculated from Kunitz's equation:

$$\psi(\varphi) = \frac{1 + 0.5 \cdot \varphi}{(1 - \varphi)^2} \tag{1}$$

for the suspension parameters:

S_N suspension density 30 kg/m^3,

ρ_{cs} density of the solid phase suspension determinated by helium method 3048 kg/m^3,

φ volume fraction 0.01m^3/m^3,

relative viscosity computed with the equation (1) was $\psi = 1.025$. While the substitution viscosity was μ = 0.001025 Pa·s.

The density of fluid was calculated as the substitution fluid density and density of the solid phase suspension as it follows:

$$\rho_z = \frac{m_{cs} + m_w}{V_z} \tag{2}$$

where:

m_{cs} mass of the solid body in 1m^3 of suspension was 30kg

m_w mass of water in 1m^3 of suspension was 988.37kg

V_z suspension volume 1m^3

Consequently, substitution density of suspension was 1018.4 kg/m^3. The above given quantities were input data parameters.

Flow parameters

Velocity of the flow in the input of the diffuser was 1.415 m/s. For that velocity, with the diameter of pipeline equal ϕ 100 mm, the flow is turbulent. Therefore, the calculation was done for the turbulent flow. For calculations, the authors used a k-epsylon model together with the specified diffuser wall roughness. It was not necessary to take the heat exchange into consideration, thus, the adiabatic flow was assumed to the calculation. Another assumption was the incompressibility of the fluid. Such an assumption was possible because the being modelled fluid had similar parameters to water.

Simulation

Simulation of the prototype diffuser work (Fig. 2) was the first stage of the research. That device was not equipped with elements to assist in steady distribution of suspension. The analysis was done to the simplified geometry. The simplification of geometry was possible because of vertical and horizontal symmetrical planes in the device. Additionally, it could be done because of fluid parameters described in a paragraph VI (without thermal calculations). The aim of the simulation was to get velocity distribution on the output of the diffuser and next, on the basis of that to do the constructional changes in the geometry to reach steady velocity distribution. The result of simulation – the magnitude velocity distribution – is presented at the Fig. 3. The graph of the *V*-velocity distribution in a horizontal symmetric plane is presented at the Fig. 4. *V*-velocity is the velocity in the direction of ordinate axis, which means axis parallel to the pipeline axis. We can notice that velocity is highly diversified. The highest velocity is in the middle of the diffuser and together with the increase of the distance from a pipeline axis the velocity is falling down dramatically and reaches oppositely directed value, and next it goes to zero value. For the proper work of device, the authors had to change construction of the device to reach steady velocity distribution in the output.

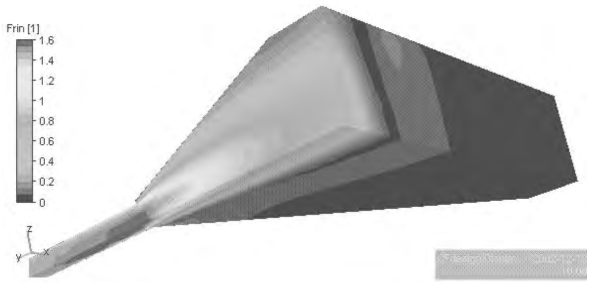


Fig. 3. Distribution of velocity magnitude in the diffuser before construction changes

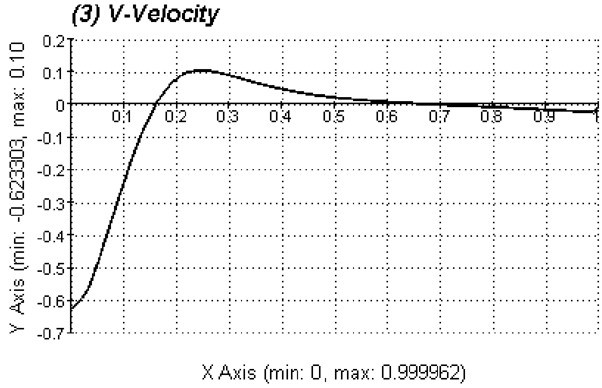


Fig. 4. Distribution of v-velocity on the output of the diffuser (on the horizontal symmetry plane) before introduction construction changes

The authors decided to another simplification because three-dimensional geometry required big computational power and computational process was too long. Thus, the authors assumed planar geometry to analysis. The velocity distribution on the symmetrical plane of diffuser was analysed because on that plane the distribution of velocity was searched for. It is a great simplification but it lets us reach the result (close to the expected) faster. We should remember that the velocity distribution in two-dimensional system reflects only to some extent the three-dimensional system.

The results of simulation on planar system are presented at the Fig. 5. We can notice that the velocity distribution along ordinate axis is similar to distribution in three-dimensional system. The difference is only in vale of velocity while the velocity profile is the same in both.

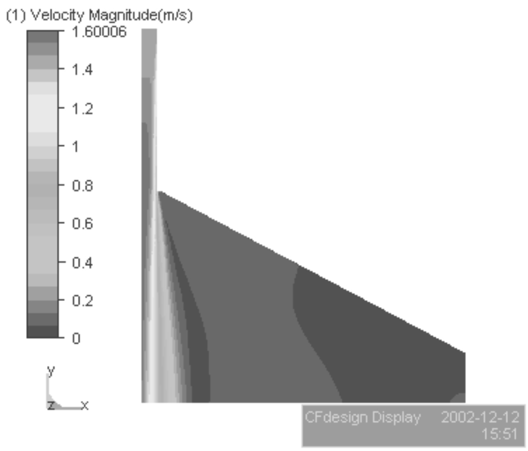


Fig. 5. Distribution of velocity magnitude before construction changes – planar system

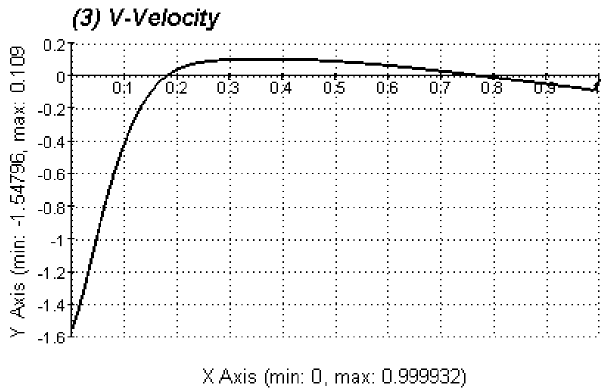


Fig. 6. Distribution of v-velocity on the output of the diffuser (on the horizontal symmetry plane) before introduction construction changes

To reach the steady velocity distribution on the output of the diffuser the authors introduced various geometrical changes. Firstly, they reduced the initial velocity by the usage of diffuser with 14° angel of flare. However, the above solution appeared to be insufficient because the velocity profile was still unsteady. Therefore, the authors placed the disturbance element in the flow axis. Consequently, they managed to decrease the flow velocity in the axis of the diffuser (Figs.7 and 8).

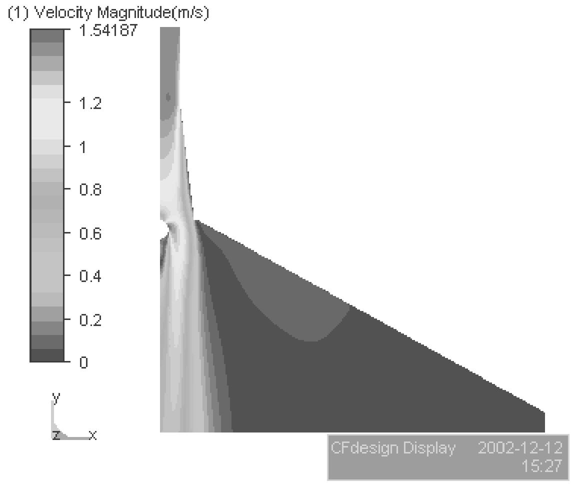


Fig. 7. Distribution of velocity magnitude in the diffuser after introduction construction changes – planar system

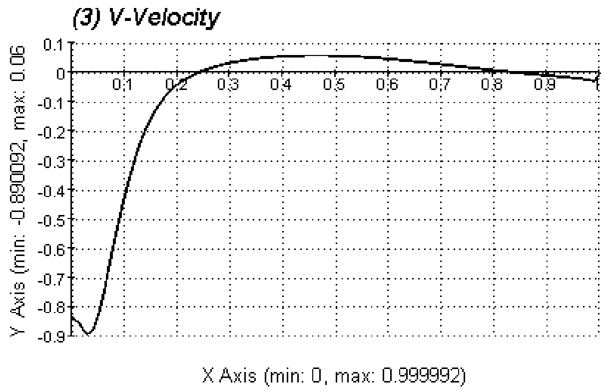
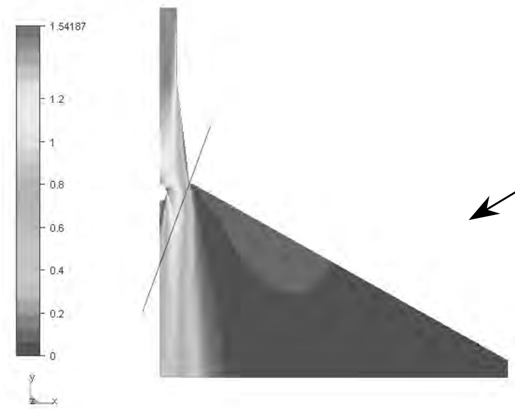


Fig. 8. Distribution of v – velocity on the output of the diffuser (on the horizontal symmetry plane) after introduction construction changes

Next step was the introduction of the stator system distributing the suspension. Five elements were applied to the diffuser. They were arranged in a specific way so as to equalise the stream in every channel. The number of stators was determined by the need of avoiding the big losses triggered by the additional bends in the flow.



The line on the crossing of the horizontal symmetry plane of the diffuser with the input plane of the stators system

Fig. 9. Location of the stator's system input plane

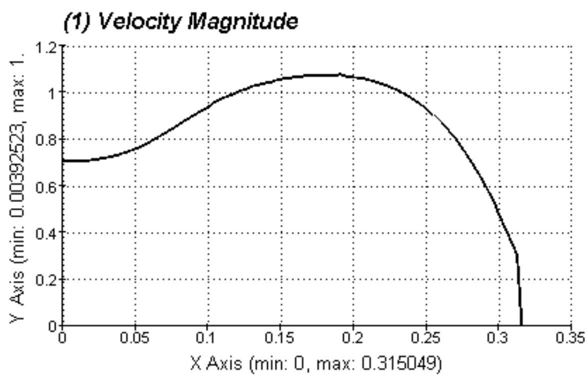


Fig. 10. Distribution of velocity magnitude along a line on the input of stators

The arrangement of stators in the diffuser underwent two conditions: the plane of the stator input (Fig. 9) was to cross the biggest flow stream (generated in earlier simulations), and the distribution of stream was to be performed on as long stream section as it was possible.

The above arrangement was performed on the basis of the velocity distribution (Fig. 10) and mathematical function describing the outline of the flow geometry along a line on the input of stators (Fig. 9).

Then, the authors delineated mathematical functions describing the velocity distribution and the change of geometrical system. The arrangement of stators was done on the basis of the following equation:

$$Q_p = \frac{\int_{x_i}^{x_{i+1}} v(x)dx}{\Delta x} \cdot \int_{x_i}^{x_{i+1}} f(x)dx \tag{3}$$

where: Q_p – stream flowing the channel created by the neighbouring stators, $f(x)$ – mathematical function describing the outline of the flow geometry along a line on the input of stators, $v(x)$ – polynomial describing the velocity distribution along the intersection, x – distance from the proper axis along proper line

Additionally, the authors introduced to the changed geometry the stators, which were eliminating the flow towards x-axis. Simulation was illustrated on Figs. 11 and 12.

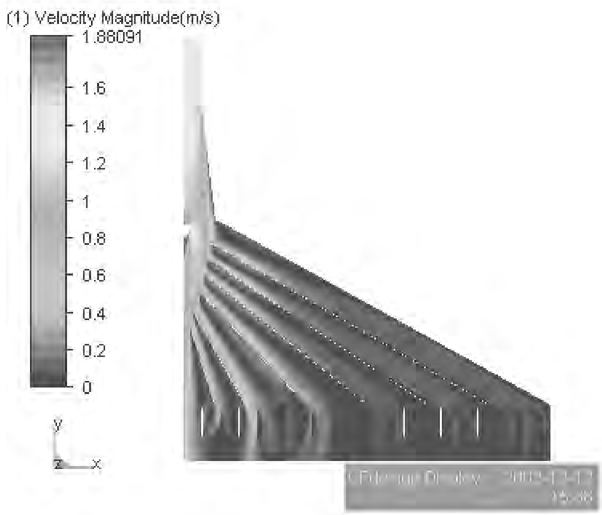


Fig. 11. Distribution of velocity magnitude in the diffuser after introduction distribution and stabilizing stators – planar system

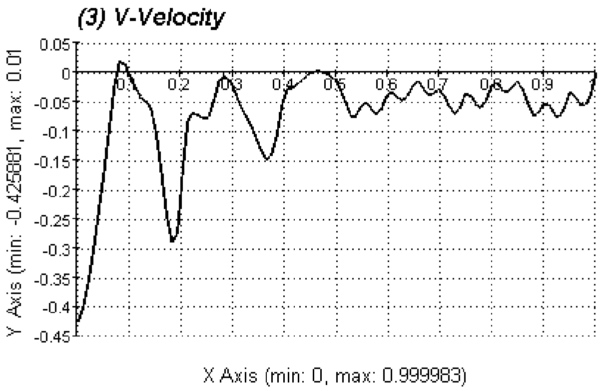


Fig. 12. Distribution of v-velocity on the output of the diffuser (on the horizontal symmetry plane) after introduction distribution and stabilizing stators

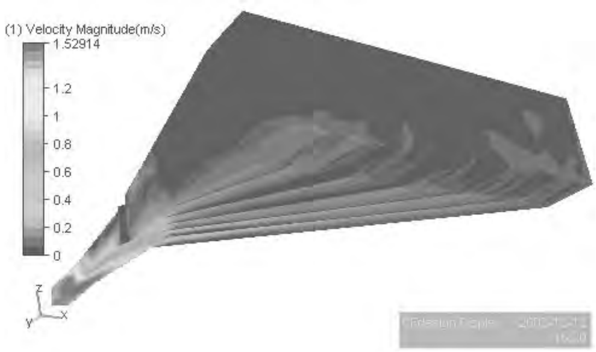


Fig. 13. Distribution of velocity magnitude in the diffuser

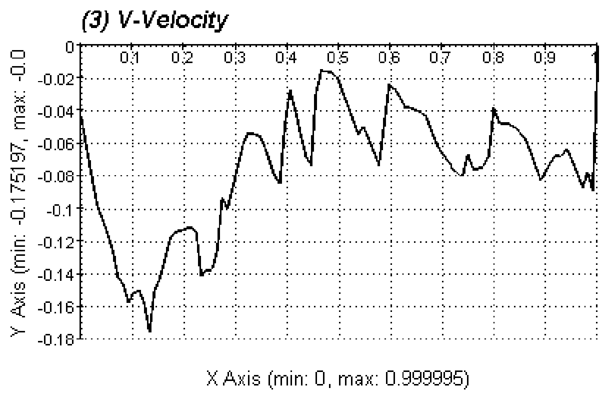


Fig. 14. Distribution of V-velocity on the output of the diffuser (on the horizontal symmetry plane)

Fig. 12 shows that the introduction of stators distributing the suspension and stabilizing the flow caused the immense improvement in the geometry in comparison to the initial one. However, the results were not fully satisfying the authors because there were ten time differences in velocities. Consequently, they decided not to introduce another improvements in the planar system (which is an extreme simplification) but to continue analysis in the three dimensional system.

After simulations on the three-dimensional geometry, the authors reached the results presented at the below Figs.

Diversification of velocity for the modified construction of the diffuser is slight in comparison to the profile of velocity in the initial diffuser construction. The maximum velocity did not exceed here 0.15 m/s. While for the initial diffuser construction, the value was 0.7 m/s. Moreover, because of the construction changes, the reverse velocity was eliminated. The new construction of the diffuser includes: initial diffuser, disturbance elements and two system stators (distribution and stabilizing stators). The above is presented at Fig. 15.

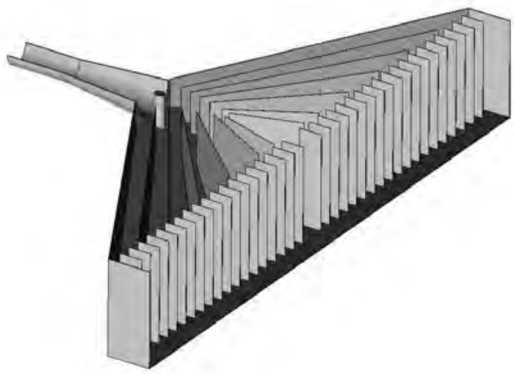


Fig. 15. Diffuser after introduction of construction changes

To sum up the article one can say that the use of a numerical, computational method in designing is not only fashionable but first of all, very helpful for the designers. It allows for visualisation of the process, which, up till now, could be observed only during experiments. It also allows for simulation, which could be very hard to make in reality for the sake of limits connect with models and real objects. Additionally, the way of geometrical modification on the grounds of CFD methods is much easier and faster than constructing real devices and their later modification.

All simplifications applied to calculations allowed for the achievement of satisfying results in short time. After the introduction of proper stators and elements influencing the shape of stream into the diffuser geometry, the authors managed to improve the velocity distribution on the output of diffuser. Therefore, we can expect that the designed device would work properly and its capabilities would be fully exploited. However, it is worth mentioning that it would not be possible with the initial flooding system.

1. Z. Orzechowski, J. Prywer, R. Zarzycki, Fluid Mechanics in Environment Protection WNT, Warszawa, 1997. 2. J. Bandrowski, H. Merta, J. Zioło, Sedimentation of Suspension. The Fundamentals and Design Wydawnictwo Politechniki Śląskiej, Gliwice, 2001. 3. T.J Chung, Finite Element Analysis in Fluid Dynamics, McGraw-Hill, Inc., 1978. 4. CFdesign Solver Technical Reference, version 5.0, Blue Ridge Numerics, Inc. 1992-2001. 5. CFdesign User's Guide, version 5.0, Blue Ridge Numerics, Inc. 1992-2001.