ZESZYTY NAUKOWE POLITECHNIKI ŚLĄSKIEJ 1990

Seria: INŻYNIERIA ŚRODOWISKA z. 33 Nr kol. 1033

Hartmut HEINEN
Manfred ZELLER

Lehrstuhl für Warmeübertragung und Klimatechnik
der RWTH Aachen

NUMERICAL CALCULATION OF LATERAL EXHAUST SYSTEM FOR OPEN VESSELS IN INDUSTRIAL APPLICATION

Summary. In recent years, mathematical models for description and prediction of air flows appear more frequently in science and technic. By use of these models in ventilation technic the hope is to conduct studies of parameters quicker and more simply and to attain a deeper understanding of the details of air flow mechanics. Presently the application of such mathematical models is still limited by the capacity of existing computers. With some success it should be possible to use these models for problems which can be simplified to two-dimensional arrangements. The mathematical model described in this paper will be used for the prediction of air flow mechanics with diffusion in closed room.

For that purpose the coupled conservation equations of mass, momentum, energy and concentration, which describe this problem, will be solved with numerical methods. The influence of turbulence on the exchange-mechanismus is analysed using the k-turbulence model with additional terms for the inclusion of buoyancy effects on turbulence.

Lateral exhaust systems are considered as a possible application of the computer program based on this theory. These are used in industrial application over dipping processes at open vessels to reduce the environmental load of harmful substances. The intention is to create a more reliable and physically well-founded basis for the planning and determination of convenient operating conditions. Initial results from the influence of bath temperature and geometry of the pull slot will be presented. In addition to these investigations, measurements will be performed on an experimental set-up to verify the validity of the mathehematical model. The possibility to use similar numerical methods will be discussed by the comparison of numerical prediction and measurements.

INTRODUCTION

In recent years, mathematical models for the description and prediction of air flows appear more frequently in science and technique in addition to experimental investigations. Caused by the increased demands on the accuracy of such models and supported by the speed in the development of greater and faster computer systems more and more complex and extensive mathematical models are going to be developed. This is also valid for research in ventilation technique. At the example of lateral exhaust systems used

in industrial application at open vessels to reduce the emission of harmful substances evaporated from the liquid surface a physical and mathematical model for the description of bouyancy affected flows in closed rooms will be presented.
The purpose of these exhaust systems is to collect and to exhaust the harmful substances (solvents etc.) - arising from the liquid surface during the manufacturing process - directly at their origin. In this way the diffusion of harmful substances into the surroundings can be prevented and the health of people will be protected. The 3 types of lateral exhaust systems used in industrial application are shown in principle in fig.1.

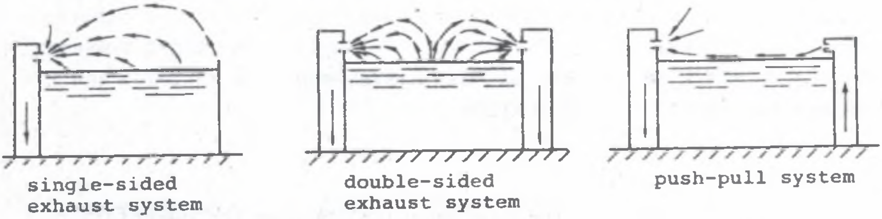


Fig. 1. Various types of lateral exhaust systems

Rys. 1. Różne rodzaje układów odciągów bocznych

There are a lot of american and german publications /1/,/2/,/3/, that deal with recommendations on the quantity of the exhaust flow rate being necessary for the total capture of the harmful substances. These recommendations based on empirical relations or determined from a small number of experimental results are deficient in several respects:
- They give no information about concentrations at points near the vessel if the recommended flow rates are satisfied; especially the maximum height of the plume of harmful substances above the vessel is not known.
- They crudely account for the influence of bouyancy effects
- The recommendations are imprecise and the tolerance is too large because of the small number of reliable experimental data.

Because of these shortcomings there exists an increasing demand to improve these design criteria. These improvements should include all parameters that affect the air-flow conditions and the diffusion of the harmful substances such as:
- exhaust flow rate
- temperature of the liquid surface
- type of liquid
- room air current (direction and velocity)
- geometry and arrangement of the exhaust slot (width of the slot, height above the liquid surface etc.)
- height of existing flanges
- bath arrangement
- blowing flow rate
- geometry of the blowing slot

Because of the large number of relevant parameters affecting the problem a systematic experimental investigation varying all these parameters is not possible. There would be too much experimental effort.
Therefore the relevant dependencies should be predicted numerically with a mathematical model describing the air-flow conditions and the phenomena of diffusion over the liquid surface and the exhaust opening.

To support the numerical results and to improve the mathematical model, if necessary, a number of experiments for specific geometric arrangements have to be carried out. The purpose of these experiments is to compare the flow , temperature and concentration fields with those obtained from calculation.
Mathematical models developed in this way can be applied to predict the distribution of harmful substances for varying geometries or all other process parameters. In this way it should be possible to predict the capture efficiency for the chosen geometric arrangement and to optimize the exhaust system already during the planning phase.

MATHEMATICAL MODEL

The numerical treatment of the problem described above is complicated by the fact, that the evaporation at the liquid surface and the bouyancy effects lead to a strong coupling of the transport mechanisms for mass, momentum, energy and concentration. Therefore a complete description of the physical processes using empirical methods (e.g. jet-laws) is not possible.
In this investigation the coupled transport equations for mass, momentum, energy and concentration are solved numerically using suitable formulations for the turbulent properties of the flow. These properties are described by a bouyancy extended k-ε-turbulence model, which includes additional terms for the influence of bouyancy effects on the production and dissipation of the turbulent kinetic energy k. The stress-flow model was applied for the determination of the additional terms.
In this way a new model arising from the exact transport equations for the turbulent correlations is developed, in which the density fluctuations itself - and not the temperature , enthalpy or concentration fluctuations - describe the effect of bouyancy on the turbulence properties. This yields a turbulence-model, that can be used more generally than the standard k-ε-model. It is able to simulate the asymmetrical influence of the bouyancy forces on the turbulent fluctuations. A more detailed description of these extensions may be found elsewhere /4/.

In spite of the development of more and more large and efficient computer-systems it is not possible to solve such problems completely, i.e. in their real 3-dimensional structure. The coupling of the individual differential equations and the influence of the bouyancy-forces require an extremely large computer memory and very long computation time. For this reason current computer systems restrict the treatment of lateral exhaust systems to 2-dim. arrangements.
Using the following assumptions

- 2-dim. steady-state flow
- cartesian coordinate system
- temperature and concentration dependend properties
- no chemical reaction
- no dissipative effects

the system of coupled differential equations can be put into the general form:

$$\frac{\partial}{\partial x}(\rho u \Phi) + \frac{\partial}{\partial y}(\rho v \Phi) = \frac{\partial}{\partial x}\left(\Gamma_\Phi \frac{\partial \Phi}{\partial x}\right) + \frac{\partial}{\partial y}\left(\Gamma_\Phi \frac{\partial \Phi}{\partial y}\right) + S_\Phi$$

convection = diffusion + source

The general variable Φ in this equation represents the velocity-components u and v, the enthalpy h, the mass-fraction ξ and the turbulent quantities k(turbulent kinetic energy) and ε (turbulent energy dissipation) whereas the variable Γ_Φ represents the diffusive transport coefficients belonging to these quantities.

The source-term S_ϕ summarizes all terms of the individual transport equations, that cannot be written as convective or diffusive terms.
In addition to the assumptions made before it is furthermore assumed, that the mixture of air and harmful substances fulfill the ideal gas law.

In solving the system of coupled, non-linear elliptic differential equations numerically a finite difference method introduced by Pun and Spalding /5/ ("Simple-Algorithmus") is used.
In the present investigation the problem is described on a grid of up to 35x35 points. The method of the staggered grid arrangement is applied. This results in 3 various grid arrangements, one for each of the velocity-components u and v and one for all other transport quantities (h, ξ, k, ε). The advantage of this staggered grid arrangement is the possibility to predict the convective fluxes through the cell boundaries directly from the field values without any interpolation.
Near the walls "wall-functions" are applied to describe the transport phenomena between the wall-next grid points and the wall itself. These "wall-functions" originally developed for an one-dimensional Couette-Flow can also be used approximately for flows with regions of recirculating eddies /4/. Using this method it is possible to reduce the number of grid lines close to the wall in spite of the large gradients in this region. This is of great importance because of the described restrictions in computer-storage. Without using this method too few grid lines remain for an adequate description of the main flow.
For the numerical solution of the finite difference equations the UPWIND-difference scheme developed for convection affected flows is applied. The significance of this scheme is a high numerical stability in connection with a satisfying behaviour of convergence. In spite of the choice of this relatively stable difference scheme for the calculations of the actual flow conditions a special underrelaxation method introduced by Neuberger /6/ has to be used. This is necessary to reach a convergent solution because of the great influence of the gravity forces.
Neuberger determined an allowed local relaxation factor in analogy to the allowed time step Δt for stability in parabolic problems using a disturbance formulation. When this local relaxation factor is not exceeded no kind of divergence can occur during the iteration process. For some of the calculations convergence of the solution could only be achieved using these locally different relaxation factors. The relaxation method is described in more detail in /4/.

The calculations of the present problems require a typical computing time of about 4000 CPU-seconds on the computer-system CYBER 175 of the RWTH Aachen Computer Centre. Typically about 3000 iterations are necessary for a convergent solution.

GEOMETRY AND PHYSICAL BOUNDARY CONDITIONS

With the methods described above the relevant transport equations can be solved for the various geometric arrangements taking into account the different thermal and flow boundary conditions. For the example of the push-pull system the calculation domain is shown in fig.2.
The push flow enters the field above the open vessel at the righthand boundary of the calculation domain. Along the liquid surface the mass fraction of harmful substances in the jet increases and at the lefthand corner the contaminated air will be

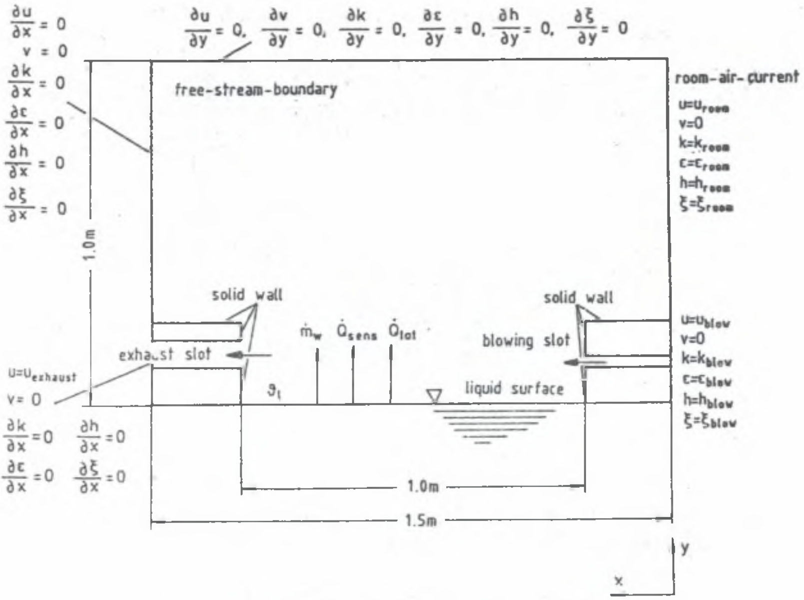


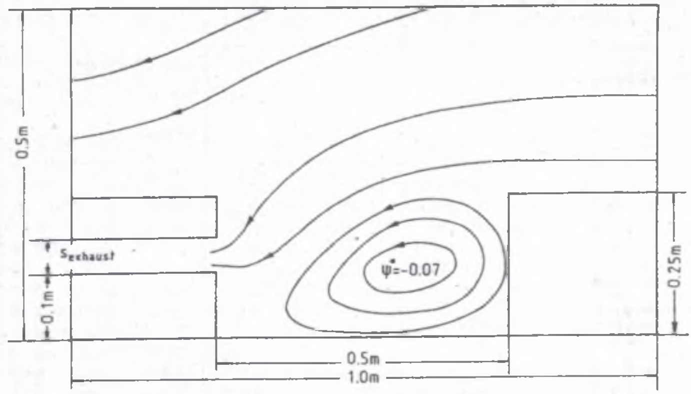
Fig. 2. Geometry and boundary conditions
Rys. 2. Geometria i warunki brzegowe

captured by the exhaust air flow. This air flow is superposed by a room air flow over the push- and pull-opening.
The liquid surface, the source of the contamination, is located at the bottom of the domain, see fig.2. The rate of evaporation at the liquid surface will be first predicted applying the analogy between heat and mass transfer. The local thermal heat transfer coefficient α at the liquid surface depends on the air conditions near the interface and may be predicted from the solution of the differential equations at this interface.
An adaption of this evaporation rate to the total amount of evaporated substances - given from experiment under the same conditions - can be carried out later without difficulties, if that seems to be nescessary. There are 4 different types of flow boundaries in the calculation domain:

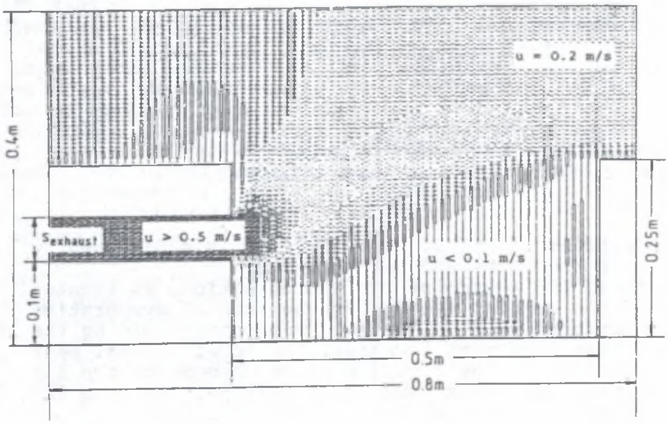
- solid walls
- liquid surfaces
- free-stream boundaries
- exhaust and blowing openings

The boundary conditions for the individual transport equations at these boundaries are also shown in fig.2.
The exhaust flow rate is given as an inlet condition for the calculation. The velocity of the exhaust flow follows from this taking into account the slot geometry. For all other field values at the exhaust opening and at all other interfaces to the environmental flow boundary conditions of the second order are assumed. Inlet conditions are assumed for all field values at the inlet boundaries (room air flow, push flow). The transport phenomena near solid walls are described by using the "wall-functions" as explained before.
As thermal boundary conditions at the solid walls both adiabatic insulation and local heat transfer can be assumed. Taking heat transfer at the wall into account the temperature of the wall has to be prescribed from experimental data.

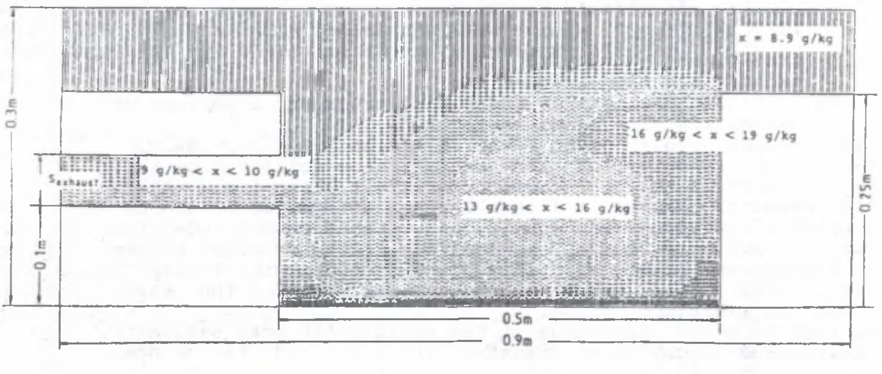
3a)


0.5m
S_exhaust
0.1m
ψ=-0.07
0.25m
0.5m
1.0m

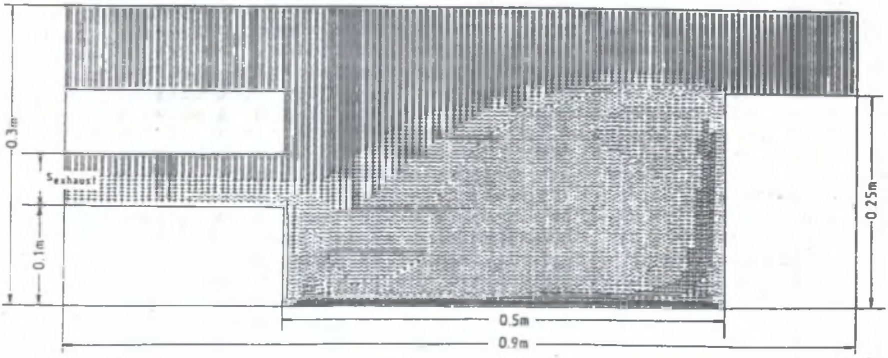
3b)


u = 0.2 m/s
0.4m
S_exhaust
u > 0.5 m/s
u < 0.1 m/s
0.25m
0.1m
0.5m
0.8m

3c)


x = 8.9 g/kg
0.3m
16 g/kg < x < 19 g/kg
S_exhaust
9 g/kg < x < 10 g/kg
0.25m
13 g/kg < x < 16 g/kg
0.1m
0.5m
0.9m

3d)



3e)

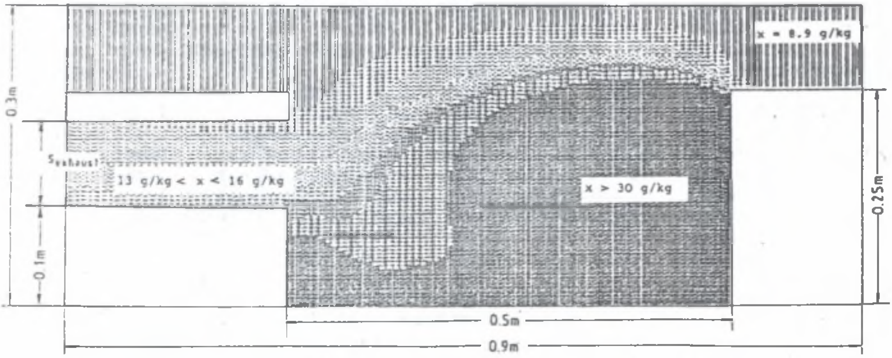


Fig. 3: a) Streamline contours
b) Velocity distribution
c) Distribution of concentration
and d) Temperature distribution
in the vicinity of an open vessel ventilated with a single-sided lateral exhaust system (partial view)
Characteristic data: $\dot{Q}_{exhaust} = 200\ m^3/h$ $\vartheta_{liquid} = 40^oC$
$u_{room} = 0.2\ m/s$ $s_{exhaust} = 0.067\ m$
e) Distribution of concentration for
$\vartheta_{liquid} = 60^oC$ $s_{exhaust} = 0.088\ m$

Rvs. 3. a) Linia prądu, b) rozkład prędkości, c) rozkład gęstości, d) rozkład temperatury. W pobliżu otwartego zbiornika wentylowanego za pomocą jednostronnego bocznego układu odciągowego (widok częściowy)
Charakterystyczne dane: $Q_{odciąg} = 200\ m^3/h$ $Q_{cieczy} = 40^oC$
$u_{pomieszcz.} = 0,2\ m/s$ $S_{odciąg.} = 0,067\ m$
e) rozkład gęstości dla $Q_{cieczy} = 60^oC$ $S_{odciąg.} = 0,088\ m$

RESULTS

Fig.3 gives first numerical results for a single-sided exhaust system showing streamline contours and graphical descriptions of the velocity , temperature and concentration fields. The graphs are printed as velocity-, temperature- and concentration classes. Symbols of higher density represent higher velocities, temperatures and concentrations, respectively. Fig.3a shows a large recirculating eddy appearing in the righthand corner of the vessel. The recirculating eddy is caused by the obstructed entrainment of air from the regions above the liquid surface (region between pull slot and liquid surface).

The air flow velocities in this region are very low in comparison to those in the region above the slot as is clearly shown in fig.3b. The major part of the total amount of exhausted air flow will be captured from the regions above the rim of the vessel.

The recirculating eddy is substantially involved in the generation of the concentration field above the liquid surface (see fig.3c). Due to this eddy the plume of harmful substances generated at the liquid surface can rise into the room and is captured by the room air current over the vessel. In this way the harmful substances reach the exhaust slot at the lefthand corner. In the actual configuration the total ammount of evaporated substances will be captured in spite of the low flow rate of exhaust air comparing with the recommendations /1/,/2/,/3/. A comparision between fig.3c and fig.3d shows a good qualitative agreement between temperature- and concentration-fields.

A first study of parameters deals with the influence of the temperature of the liquid surface on capture efficiency and air-flow conditions. The major effect of the increased bath temperature is an increase of the total amount of evaporated substances as shown in fig.3e (see the large region of high concentrations in the righthand corner). An increase in bath temperature of about 10 K leads to an increase of the evaporated amount of about 100%. All other conditions were the same in both cases. The influence of the bath temperature on the maximum height of the plume is very low (see fig.3e and 3c) as well as the influence on the velocity field above the vessel. Only the bouyancy-affected velocities at the righthand corner of the vessel increase a little bit.

Because of the increasing height of the plume under these conditions cross drafts, which cannot be investigated under 2-dim. conditions, may prevent a total capture of the plume.

For the configuration under consideration and for a bath temperature of about 60 °C total capture cannot be obtained with an exhaust flow rate of 200 m^3/h.

Another study deals with the influence of the geometric arrangement of the exhaust opening.

There was no significant influence of both the shape of the exhaust opening (sharp or well rounded edges) and the velocity profile assumed in the opening on the air-flow conditions, on the evaporation rate, and on the capture efficiency. Only the velocities in the region near the opening change a little . The major reason for this phenomenon is the limited domain of suction-systems.

Changes in the width of the exhaust opening leads to similar results. In spite of reducing the width of the slot from 0.106m down to 0.053m (corresponding an increase in the velocity of the exhaust slot from 0.5 up to 1.0 m/s) there were no significant changes regard to the velocity and concentration fields.

For both the test of the mathematical model and for the check of 3-dim. effects, which cannot be investigated with the mathematical model, experimental investigations will be carried out in the near future. With the experiences made in similar investigations /7/ a good qualitative and with restrictions also quantitative agreement between measurement and prediction may be expected as long as the air flow conditions are approximately two-dimensional. Quantitative disagreement can only be produced by the numerical diffusion on the predicted flow field and by the bouyancy effects, which generate 3-dim. effects in the experiments.

An important progress would be obtained if the development of computer systems made it possible to investigate the problems in their real 3-dim. structure.

REFERENCES

[1] ACGIH, 1978 "Industrial ventilation, a manual of recommended practice", 15th ed. Lansing, MI: American Conference of Governmental Industrial Hygienists

[2] ASHRAE, 1980 ASHRAE handbook-1980 systems, chap.22. New York: American Society of Heating, Refrigerating and Air-Conditioning engineers, Inc.

[3] Recknagel-Sprenger Taschenbuch für Heizung- und Klimatechnik, R. Oldenburg Verlag, 1983/84

[4] Schmitz, R.M. Berechnung turbulenter Raumluftströmungen bei gekoppeltem Impuls-, Wärme- und Stoffaustausch, Dissertation RWTH Aachen

[5] Pun, W.M., Spalding, D.B. A General Computer Program for Two-Dimensional Elliptic Flow. HTS Report 76/2, Imp. College of Sci. and Techn., London, Mech. Engng. Dep., August 1977

[6] Neuberger, A.W., Chutwani, A.U., Eickhoff, H., Kopmann,J. Selection of Relaxation Factors for Computing Steady State Turbulent Flow, Proc. Third Int. Conf. non Numerical Methods in Laminar and Turbulent Flow, Seattle (1983) 280/289

[7] Ahl, J.P. Untersuchung der Raumluftströmung bei Belüftung durch Fensterblasanlagen, Dissertation RWTH Aachen 1987

OBLICZENIA NUMERYCZNE UKŁADÓW BOCZNYCH NAWIEWNIKÓW DLA NACZYŃ OTWARTYCH W ZASTOSOWANIACH PRZEMYSŁOWYCH

Streszczenie

Modele matematyczne służące do opisywania i przewidywania przepływów powietrza pojawiają się ostatnio coraz częściej w nauce i technice. W technice wentylacji modele te stosuje się w celu przeprowadzenia badań parametrów w sposób szybszy i prostszy, a także aby umożliwić lepsze zrozumienie szczegółów mechaniki przepływów powietrza. Ciągle jeszcze zastosowanie takich modeli matematycznych ograniczone jest pojemnością istniejących obecnie komputerów. Powinno być możliwe stosowanie tych modeli dla problemów, które mogą być uproszczone do dwuwymiarowych. Model matematyczny opisany w referacie użyty zostanie do programowania mechaniki przepływu powietrza przy dyfuzji w pomieszczeniach zamkniętych. W tym celu zostanie metodami numerycznymi rozwiązany układ równań zachowania masy, pędu, energii i stężenia, które opisują ten problem. Przeanalizowany zostanie wpływ turbulencji na mechanizmy wymiany z zastosowaniem modelu turbulencji k-ε z dodatkowymi składnikami uwzględniającymi oddziaływania wyporu na turbulencję.

Rozważane są boczne układy nawiewników jako możliwe zastosowanie programu komputerowego opartego na tej teorii. Układy te używane są w zastosowaniach przemysłowych w procesach zanurzania w zbiornikach otwartych w celu zmniejszenia zanieczyszczeń środowiska szkodliwymi substancjami. Dąży się do stworzenia bardziej wiarygodnych i fizycznie uzasadnionych podstaw dla planowania i wyznaczania odpowiednich warunków pracy. Przedstawione zostaną wstępne wyniki badań wpływu temperatury kąpieli i geometrii szczeliny odciągowej. Jako uzupełnienie tych badań przeprowadzone zostaną pomiary na stanowisku doświadczalnym w celu zweryfikowania słuszności modelu matematycznego.

Przedyskutowana zostanie możliwość użycia podobnych metod numerycznych przez porównanie wyników programowania numerycznego i pomiarów.

ЧИСЛЕННЫЕ РАСЧЕТЫ СИСТЕМ, БОКОВЫХ ВЕНТИЛЯТОРОВ ДЛЯ ОТКРЫТЫХ СОСУДОВ В ПРОМЫШЛЕННОМ ПРИМЕНЕНИИ

Резюме

В последнее время в науке и технике все чаще появляются математические модели для описания и предсказывания протекания воздуха. В технике вентиляции эти модели применяют с целью более быстрого и простого исследования параметров, а также чтобы облегчить лучшее понимание подробностей механики протекания воздуха, Применение таких математических моделей ограничивается емкостью существующих компьютеров. Эти модели должны приняться для проблем, которые можно упростить до двухмерных. Описанная в докладе математическая модель применится для программирования механики течения воздуха при диффузии в замкнутых помещениях. Для этого с помощью численных методов будет решена система уравнений сохранения массы, количества движения, энергии и концентрации, которые описывают эту проблему. Будет также проанализировано влияние турбулентиости на механизмы обмена с применением модели турбулентности к - с добавочными составными элементами учитывающими воздействие гидростатического подпора на турбулентность.

Рассматриваются боковые системы вентиляторов для которых является возможным применение компьютерной программы, основанной на этой теории. Системы эти нашли промышленное применение в процессах погружения в открытых резервуарах с целью уменьшить загрязнение среды вредными веществами. Стремятся к созданию более достоверных и физически обоснованных оснований для планирования и определения соответствующих условий работы. Будут представлены предварительные результаты исследований влияния температуры ванны и геометрии оттяжной щели. Дополнительно к этим исследованиям проведутся измерения на испытательном стенде, чтобы проверить правильность математической модели. Обсудится возможность применения похожих численных методов, сравнивая результаты численного программирования и результаты измерений.