POLSKA AKADEMIA NAUK

INSTYTUT MASZYN PRZEPŁYWOWYCH

TRANSACTIONS OF THE INSTITUTE OF FLUID-FLOW MACHINERY

PRACE INSTYTUTU MASZYN PRZEPŁYWOWYCH

104

WYDAWNICTWO IMP PAN

GDAŃSK 1998

THE TRANSACTIONS OF THE INSTITUTE OF FLUID-FLOW MACHINERY

exist for the publication of theoretical and experimental investigations of all aspects of the mechanics and thermodynamics of fluid-flow with special reference to fluid-flow machines

*

PRACE INSTYTUTU MASZYN PRZEPŁYWOWYCH

poświęcone są publikacjom naukowym z zakresu teorii i badań doświadczalnych w dziedzinie mechaniki i termodynamiki przepływów, ze szczególnym uwzględnieniem problematyki maszyn przepływowych

Wydanie publikacji zostało dofinansowane przez PAN ze środków DOT uzyskanych z Komitetu Badań Naukowych

EDITORIAL OFFICE – REDAKCJA

Wydawnictwo Instytutu Maszyn Przepływowych
Polskiej Akademii Nauk
ul. Gen. Józefa Fiszera 14, 80-952 Gdańsk, skr. poczt. 621,
☎ (0-58) 341-12-71 wew. 141, fax: (0-58) 341-61-44,
e-mail: esli@imppan.imp.pg.gda.pl

ISSN 0079-3205

TRANSACTIONS OF THE INSTITUTE OF FLUID-FLOW MACHINERY

No. 104, 1998, 31-44

BOZIN DONEVSKI[1], IGOR NEDELKOVSKI[1], CVETE STEFANOVSKA[1]
KAZIMIERZ BRODOWICZ[2], KRZYSZTOF OSTROWSKI,[2] GOCE NIKOLOVSKI[3]

Numerical computation of steam flow in power plant condensers utilizing the three-dimensional procedure

One of the salient shortcomings of the most present computational methods for predicting the steam flow in the power plant condensers is the limitation of two-dimensional or quasi three-dimensional flows. However, the flow in the untubed region of condensers (from turbine exhaust to the outer circumference of the tube bundle) is markedly three-dimensional, and the distribution of steam around the tube bundle is made even more three-dimensional because of the temperature difference caused by the cooling water. The steam flow in the axial direction in the tube bundle region, is constrained by supporting plates. However, the three-dimensional modeling is necessary and important in providing an accurate prediction of the flow, especially in the regions of air suction. A practical approach is thus needed to establish an algorithm, in which included are the three-dimensional effects in a real and practical manner, in both the untubed and tubular regions in the condensers of the power plant.

A full description of the three-dimensional procedure for a numerical computation of the steam flow in power plant condensers is presented in this paper. The purpose of the present study is to develop an algorithm that can be used to predict the nature of the three-dimensional fluid flow and heat transfer in large condensers of power plants. In order to demonstrate the applicability and predictive capability of the proposed method, both the three-dimensional and two-dimensional procedures are applied to simulate velocity, pressure and fraction of air mass experimental condensers. Numerical results obtained are compared with the experimental results of these condensers, as they appear in the scientific relevant literature on numerical simulation.

Nomenclature

a – heat transfer area per unit volume, m^2/m^3
c_g – air mass fraction $= \rho_g/\rho$
d_i – inner diameter of tube, m
d_o – outer diameter of tube, m
D – molecular diffusion coefficient, m^2/s
D_p – coefficient vapour diffusion in air, $kg/(Pa \cdot m \cdot s)$
F – flow resistance forces, N/m^3
f – friction factor
k – overall heat transfer coefficient, W/m^2K
T – temperature, K
u – velocity component in the x direction, m/s
v – velocity component in the y direction, m/s
w – velocity component in the z direction, m/s
α – heat transfer coefficient, W/m^2K
ε – local volume porosity
λ – thermal conductivity, W/mK

[1]Faculty of Technical Sciences, University of „St. Kliment Ohridski" 97000 Bitola, Macedonia
[2]Institute of Heat Engineering, Warsaw Technical University, Nowowiejska 25, 00-665 Warsaw
[3]Power Plants REK-Bitola, Electric Power Company of Macedonia, 97000 Bitola, Macedonia

r	–	latent heat of condensation, J/kg	μ	–	molecular viscosity, kg/ms
m	–	condensation rate per unit volume, kg/m^{3}s	μ_{eff}	–	effective viscosity, kg/ms
n	–	number of horizontal tubes in a vertical row	μ_t	–	turbulent viscosity, kg/ms
p	–	pressure, Pa	Θ	–	correction factors
s	–	tube pitch, m	ρ	–	density, kg/m^3
q	–	heat flux per unit area, kW/m^2	ξ	–	resistance coefficient, 1/m
R	–	flow resistance; gas constant			

Dimensionless criteria

E_g	–	p_g/p	R_p	=	$(\rho_k\mu_k/\rho_p\mu_p)^{0.5}$
Fr	–	Froud number	Re	–	Reynolds number
F	=	$Pr_k/(FrH)$	Re_f	=	$2\pi d_o q/(\mu_k r)$
G	=	$Rp \cdot H/Pr_k$	Re_q	=	$Re_f + Re_k(\rho_p/\rho_k)^{0,5}$
H	=	$c_{pk}(T_{pk} - T_c)/r$	Z	=	$(1+1/G)^{2/3}/F^{0,5}$
Pr	–	Prandtl number			

Subscripts

c	–	tube	o	–	outer diameter
g	–	gas	p	–	steam
i	–	inner diameter	pk	–	interface (steam-condens.) surface
k	–	condensate	w	–	cooling water

1. Introduction

There are numerous papers and reports investigating and describing the numerical simulation of steam flow and heat transfer in the condensers of power plants. The simulations, as presented in most of the literature on studies conducted, are mainly based on the two-dimensional mathematical model or its quasi-three-dimensional modification, Zhang and Bokil (1997) who define the steam flow in the tube bundle as a flow through porous media. A general characteristic of these models on which they are based, is the assumption that the flow in the power condensers can be examined as a two-dimensional flow. This assumption in some papers, as presented by Brickell (1981), Marto (1984) and Butterworth (1994), related to the problem of the simulation of heat - flow processes in the power plant condensers, is as yet inconclusive and subject to further investigation.

In the tubular region, the definition of two-dimensional models presumes that the flow of the steam is two-dimensional because of the existence of support plates along of the bundle which obstruct the flow in axial direction. However, because of the non-uniform temperature distribution of the cooling water along the tubes, different conditions for flow as well as in the axial direction exist. This was also the reason for the appearance of the quasi three-dimensional models in which the influence of temperature changes along the tube with the cooling water is taken indirectly. The three-dimensional effects are also caused by the apparatus for sucking the non-condensable gases (air). Namely, this apparatus for drawing in the air sucking must be located in a place of lower pressure, which in the power condensers, is in the vicinity of the water cooling inlet. Therefore, the sucking of the air, which is a continuous process in the direction of the sucking apparatus, causes the three-dimensional effects. Those effects in the existing two and quasi three-dimensional models, cannot be taken into consideration, because it is assumed that the air is sucked out of the surface where the calculation is postulated.

The flow in the tubular region is not the only one reason that has an influence on the performance of condensers; the flow outside the tubular region (from the inlet into the condenser to the circumference of the tube bundle) has also an influence on the effective work of the power plant condensers. It is important for the proper determination of the flow in this region, because the velocity and pressure distribution in this space have a significant influence on the velocities and pressure distribution on the outer circumference of the tube bundle and also on the flow in its region. When, steps will be taken to ensure that in this space no transversal obstacles in the axial direction exist, then it is clear that the flow outside of the tubular region is three-dimensional. This problem is one of the most significant shortcomings in the existing models that are limited to the two dimensional flow, Donevski et al., (1997).

2. Mathematical modelling

The steam flow and heat transfer are modeled, on the assumption that this condensate has a negligible momentum and occupies almost no volume, by the three-dimensional mathematical model for flow in the porous media. The steam is assumed to be saturated.

Mass conservation equation:

$$\frac{\partial(\varepsilon\rho u)}{\partial x}+\frac{\partial(\varepsilon\rho v)}{\partial y}+\frac{\partial(\varepsilon\rho w)}{\partial z}=-m. \tag{1}$$

Momentum conservation equation:

$$\begin{aligned}\frac{\partial(\varepsilon\rho u^2)}{\partial x}+\frac{\partial(\varepsilon\rho uv)}{\partial y}+\frac{\partial(\varepsilon\rho uw)}{\partial z}&=\frac{\partial}{\partial x}\left(\varepsilon\mu_{eff}\frac{\partial u}{\partial x}\right)+\\ &+\frac{\partial}{\partial y}\left(\varepsilon\mu_{eff}\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial z}\left(\varepsilon\mu_{eff}\frac{\partial u}{\partial z}\right)-\varepsilon\frac{\partial p}{\partial x}-mu-F_x,\end{aligned} \tag{2}$$

$$\begin{aligned}\frac{\partial(\varepsilon\rho uv)}{\partial x}+\frac{\partial(\varepsilon\rho v^2)}{\partial y}+\frac{\partial(\varepsilon\rho vw)}{\partial z}&=\frac{\partial}{\partial x}\left(\varepsilon\mu_{eff}\frac{\partial v}{\partial x}\right)+\\ &+\frac{\partial}{\partial y}\left(\varepsilon\mu_{eff}\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\varepsilon\mu_{eff}\frac{\partial v}{\partial z}\right)-\varepsilon\frac{\partial p}{\partial y}-mv-F_y,\end{aligned} \tag{3}$$

$$\frac{\partial(\varepsilon\rho uw)}{\partial x}+\frac{\partial(\varepsilon\rho vw)}{\partial y}+\frac{\partial(\varepsilon\rho w^2)}{\partial z}=\frac{\partial}{\partial x}\left(\varepsilon\mu_{eff}\frac{\partial w}{\partial x}\right)+$$
$$+\frac{\partial}{\partial y}\left(\varepsilon\mu_{eff}\frac{\partial w}{\partial y}\right)+\frac{\partial}{\partial z}\left(\varepsilon\mu_{eff}\frac{\partial w}{\partial z}\right)-\varepsilon\frac{\partial p}{\partial z}-mw-F_z. \quad (4)$$

Conservation of air mass fraction:

$$\frac{\partial(\varepsilon\rho uc_g)}{\partial x}+\frac{\partial(\varepsilon\rho vc_g)}{\partial y}+\frac{\partial(\varepsilon\rho wc_g)}{\partial z}=\frac{\partial}{\partial x}\left(\varepsilon\rho D\frac{\partial c_g}{\partial x}\right)+$$
$$+\frac{\partial}{\partial y}\left(\varepsilon\rho D\frac{\partial c_g}{\partial y}\right)+\frac{\partial}{\partial z}\left(\varepsilon\rho D\frac{\partial c_g}{\partial z}\right) \quad (5)$$

where the dependent variables are: velocity components u, v and p, and the air mass fraction c_g.

The constitutive equations related to other parameters which are present in the mathematical modelling are defined by the following relations:

Local volume porosity ε:

- for staggered arrangement of the tubes in the bundle:

$$\varepsilon=1-\frac{\pi}{2\sqrt{3}}\left(\frac{d_o}{s}\right)^2, \quad (6)$$

- for in-line arrangement of the tubes in the bundle:

$$\varepsilon=1-\frac{\pi}{4}\left(\frac{d_o}{s}\right)^2. \quad (7)$$

The density of mixture is determined on the basis of the equation of state, where steam and air are treated as an ideal gases; namely,

$$\rho=\frac{p}{RT} \quad (8)$$

where T is the saturation temperature determined by the partial steam pressure. Effective viscosity μ_{eff} is defined as a sum of molecular and turbulent viscosities:

$$\mu_{eff}=\mu+\mu_t. \quad (9)$$

Molecular viscosity μ is defined as:

$$\mu=\frac{(1-E_g)\mu_p+1.61E_g\mu_g}{1+0.61E_g}. \quad (10)$$

Turbulent viscosity μ_t is defined as $\mu_t=20\cdot\mu$ according to the recommendation of Zhang et al. (1991).

The fluid flow resistance forces are determined by Darcy law of linear flow, through porous media

$$F_x = \mu R_x u,$$
$$F_y = \mu R_y v,$$
$$F_z = 0.$$

where the flow resistance R_x and R_y are determined by the adequate empirical relations. The equations proposed here are the approximated experimental results for local flow resistance of a two-phase flow across the tube bundle by Brodowicz and Czaplicki (1989)

$$R_x = d_o^{-2} G \cdot f \cdot \Theta_x,$$
$$R_y = d_o^{-2} G \cdot f \cdot \Theta_y.$$

where G is a coefficient which takes into account the influence of a tube bundle geometry, f is the fraction factor, Q_x and Q_y are the correction factors which take into account the influence of the rate of condensation i.e., a two-phase flow on the flow resistance forces. These coefficients are functions of the condensation rate m, Reynolds number and the direction of the flow.
The rate of steam condensation per unit volume is determined by the equation:

$$m = a\frac{k(T - T_w)}{r} \tag{11}$$

where a is the heat transfer area per unit volume, T is the local steam temperature and T_w is the local cooling water temperature inside the tubes of the condenser

$$a = \frac{4}{d_o}\frac{1-\varepsilon}{\varepsilon}. \tag{12}$$

The overall heat transfer coefficient is determined as a sum of individual heat transfer coefficients:

$$k = \frac{1}{\frac{1}{\alpha_w}\frac{d_o}{d_i} + \frac{1}{\alpha_c} + \frac{1}{\alpha_k} + \frac{1}{\alpha_g}}. \tag{13}$$

The heat transfer coefficient from the water side, α_w is determined from the well known Mc-Adams equation for forced convection in the circular tube:

$$\alpha_w = 0,023\frac{\lambda_w}{d_i} Re_w^{0,8} Pr_w^{0,4}. \tag{14}$$

α_c is an equivalent heat transfer coefficient across the tube wall:

$$\alpha_c = \frac{2\lambda_c}{d_o \ln(d_o/d_i)}. \tag{15}$$

α_k is the heat transfer coefficient across the film of condensate, which is determined on the basis of the equation of Honda et al. (1986), for the condensation on the horizontal inundated tube in the tube bundle, while the influence of the condensate inundation is taken via the relation of Cippolone et al., (1983)

$$Nu = (Nu_N^4 + Nu_F^4)^{0,25},$$

$$Nu_N = 0,728 \cdot F^{0,25}(1 + Z + 0,57 \cdot Z^2)^{0,25} Re_k^{0,5} Nu_F = 0,11 \cdot Re_q^{0,8} Pr_k^{0,4},$$

$$Nu = Nu_0[n^{7/8} - (n-1)^{7/8}].$$

α_g is the equivalent heat transfer coefficient across the film of non-condensable gases; it is evaluated via Berman and Fuks (1958), empirical expression for the mass transfer coefficient in the tube bundle during the downward flow of the steam-air mixture:

$$\alpha_g = a \frac{D_p}{d_o} Re^{1/2} E_g^{-0.6} p^{1/3} \left(\frac{\rho_p r}{T}\right)^{2/3} \frac{r}{(T - T_{pk})^{1/3}}. \tag{16}$$

The diffusion coefficient for the steam in the air D_p is determined as:

$$D_p = D/R_p T. \tag{17}$$

D is the molecular diffusion coefficient for the steam-air mixture; it is determined on the basis of empirical equation as:

$$D = 0,00011756552 \cdot \frac{T^{1,75}}{p}. \tag{18}$$

Boundary conditions Boundary conditions are specified for the following: the boundary of the flow space; the inlet of supporting plates; and the plates for directions of the flow of steam air mixture and drain the condensate as follows

1. At the inlet of the condenser, the boundary condition is the steam velocity which is determined on the basis of the steam flow through the turbine exhaust and the cross sectional area of the inlet of the condenser.
2. On the walls of the condenser and the plates for the direction of the flow of the steam-air mixture and the draining of the condensate, the boundary condition is the normal component of steam-air mixture velocity to be equal to zero.
3. At the outlet of the condenser, the boundary condition is the outlet velocity of the steam-air mixture, which is calculated on the basis of the characteristics of the sucking apparatus.

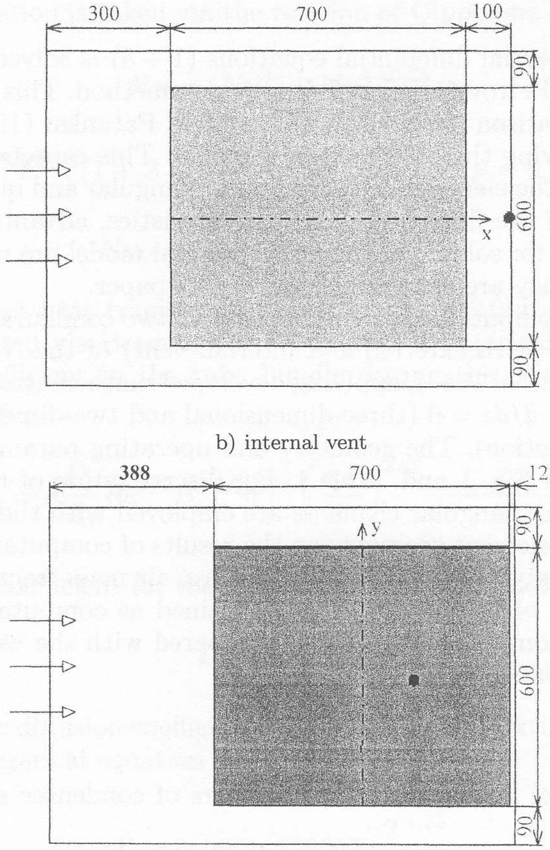
3. Mathematical model solving

The system of partial differential equations ($1 \div 5$) is solved by applying the Streamline Upwind Petrov-Galerkin finite element method. This method has some advantages in comparison to the SIMPLE method, Patankar (1980) which is usually applied for solving this mathematical model. This especially relates to the possibility of using the elements with irregular triangular and quadrilateral forms for discretization of the flow area. The characteristics, advantages and lacks of the applied method for solving of the mathematical model are not discussed here in detail, because they are not the subject of this paper.

The numerical computations are performed on two configurations of the experimental condenser (with external and internal vent) of the NEI-Parsons, Ltd., England (Al-Sanea et. al 1983), applying developed mathematical models in three and two dimensions $d/dz = 0$ (three-dimensional and two-dimensional models in the following description). The geometry and operating parameters of this condenser are shown in Fig. 1 and Table 1. For discretization of the flow area, the regular and uniform triangular elements are employed with the aim to minimize the influence of the element geometry on the results of computation (Fig. 2). The distribution of the mixture velocity and pressure, air mass fraction and heat flux in the cross section of the condenser are obtained as computational results. In Figs. 3 and 4, the computed heat flux is compared with the well proven experimental results (Al-Sanea et al. 1983).

Table 1. Geometrical and operating parameters of condenser at NEI (Al-Sanea et al. 1983)

Tube bundle		
Tube out side diameter [mm]	25.4	
Tube wall thickness [mm]	1.25	
Tube pitch [mm]	34.9	
Tube length [m]	1.219	
Number of tubes	400	
Arrangement of tubes	triangular	
Operating conditions	Internal Vent	External Vent
Pressure [kPa]	27.67	13.60
Steam inflow [kg/s]	2.032	0.792
Air inflow [kg/s]	2.48×10^{-4}	2.69×10^{-4}
Mixture outflow [kg/s]	1.097×10^{-2}	2.87×10^{-2}
Cooling water inlet temperature [°C]	17.8	13.1
Cooling water velocity [m/s]	1.19	1.18



b) internal vent

a) external vent

Fig. 1. Configuration of experimental condenser at NEI (Al-Sanea et al. 1983).

4. Analysis of the results

From the analysis of Figs. 3 and 4, it can be noted that there is not much difference between the various computed results. One can conclude that the three-dimensional model does not give much significant improvement to the results. However, the detailed analysis of the proposed model will indicate that this is the case. In fact, in the definition of the conditions of the boundary, the determined boundary condition on the inlet to the condenser is the inlet velocity of the steam-air mixture, which is calculated on the basis of the rate of steam flow in the inlet of the condenser. At this, as per this principle, the distribution of velocities in the cross section flow is defined uniformly. This assumption does not response to the conditions which are dominant in the power plant condensers,

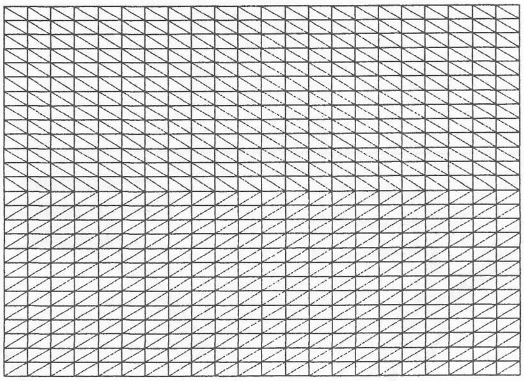


Fig. 2. Triangulation of the condenser of NEI (Al-Sanea et al. 1983).

considering the fact that in the condensers, the steam flows directly from the last turbine stages. Thus, the flow at the inlet of the condenser in no way can be uniform, but it is represented by a fully turbulent three-dimensional flow.

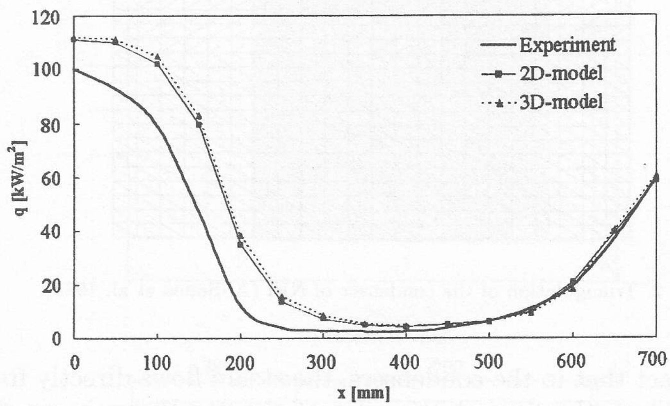
An example of the experimentally determined flowing picture for the turbine exhaust – condenser inlet, is shown in Fig. 5 (Dawidson and Rowe 1981), where the contour lines present the ratio between measured velocities and the average velocity.

When such a defined boundary condition for the inlet velocity will be used for the simulation of steam flow and heat transfer in the experimental condenser of NEI (Al-Sanea et al. 1983), the situation will be completely changed. This is confirmed by simulations performed with the arbitrarily assumed nonuniform inlet velocity as a boundary condition where significant changes of the flowing picture in the condenser can be noted.

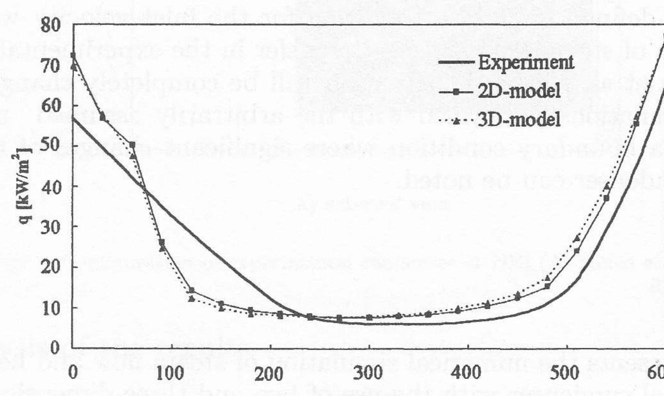
5. Conclusions

This paper presents the numerical simulation of steam flow and heat transfer in an experimental condenser with the use of two and three-dimensional mathematical models. A comparative analysis of the calculated results by using the two and three-dimensional mathematical models and the experimental results, as published in the general adequate literature, are also described. On the basis of the conducted studies, it can be concluded that the three-dimensional numerical simulation in general does not show any significant advantages compared with the two-dimensional or quasi three-dimensional simulation.

These results are achieved by adopting the uniform distribution of the velocity of steam-air mixture at the inlet of the condenser at the defined boundary conditions. However, the advantages of the three-dimensional model are seen if, in the definition of the boundary conditions, the inlet velocity of the steam-air

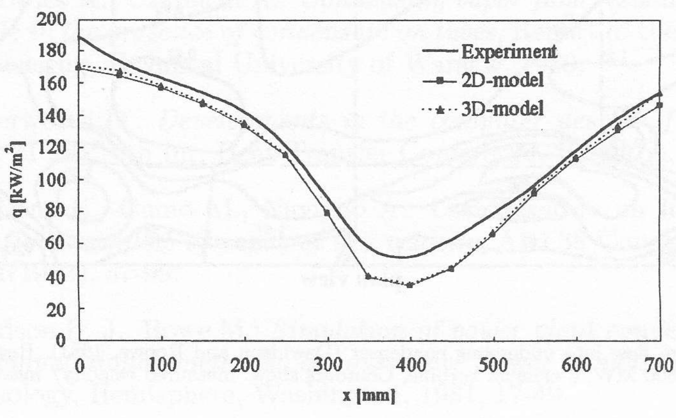


a) horizontal center line

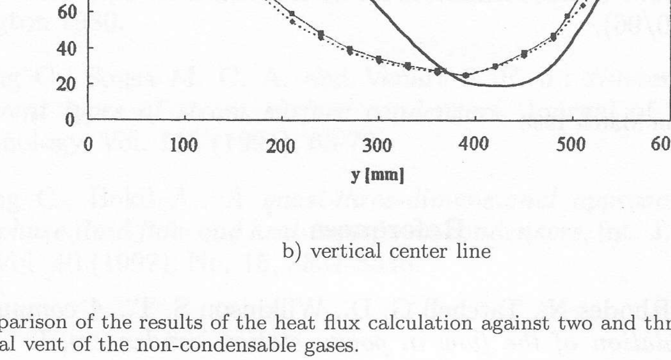


b) vertical center line

Fig. 3. Comparison of the results of the heat flux calculation against two and three dimensional models with a external vent of the non-condensable gases.



a) horizontal center line



b) vertical center line

Fig. 4. Comparison of the results of the heat flux calculation against two and three dimensional models with a central vent of the non-condensable gases.

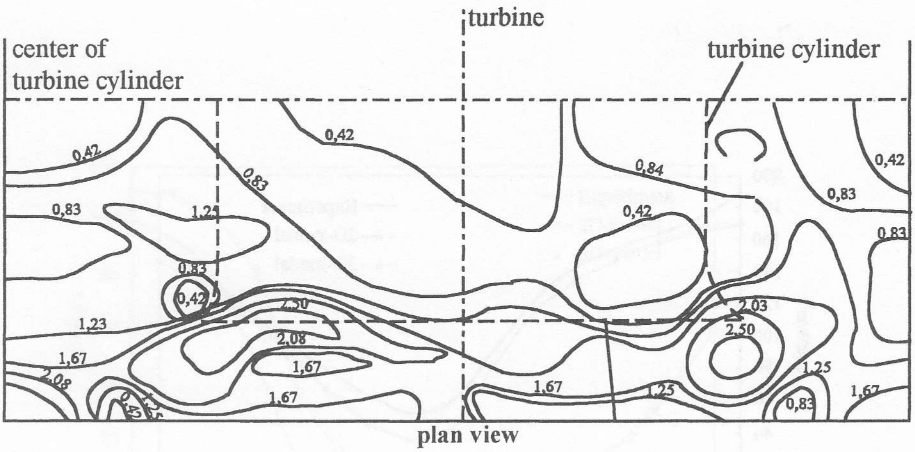


Fig. 5. Measured steam flow into undersling condenser (Dawidson and Ropwe, 1981). Region covered: below and exhaust of 500 MW, 6 exhaust turbine, Contours show: measured velocity/ mean velocity.

mixture is postulated as non-uniform. Such a defined boundary condition actually corresponds to the real conditions in the power plant condensers. This influences the future development of the method for numerical simulation of the flow in the turbine exhaust region. There is a need to develop an adequate method for the simulation of the flow in the exhaust of the turbine. The results obtained from this method could be utilized as input parameters for the numerical simulation of the flow and heat transfer in the power plant condensers.

Acknowledgement Financial support for this work was received from a joint project by both the Ministry of Science, Republic of Macedonia (Research Grant No. 40026193), and by the State Committee for Scientific Research, Poland (Research Grant No. 141/090/96).

Manuscript received in March 1998

References

[1] Al-Sanea S., Rhodes N., Tatchell G. D., Wilkinson S. T.: *A computer model detailed calculation of the flow in power station condensers*, I. CHEM. E. Symposium series No. 75 (1983), 70-88.

[2] Berman L. D., Fuks L. D.: *Mass transfer in condensers with horizontal tube bundles at presence of condensate*, Teploenergetika, No. 8 (1958), 66-74.

[3] Brickell M. G.: *Potential problem areas in simulating condenser performances*, Power Condenser heat Transfer Technology, Hemisphere, Washington 1981, 51-61.

[4] Brodowicz K., Czaplicki A.: *Condensing vapor flow resistance through tubes bundle in the presence of condensate on tubes*, Report of the Institute of Heat Engineering, Technical University of Warsaw, 1989.

[5] Butterworth D.: *Developments in the computer design of heat exchangers*, Proc. of the 10th Int. Heat Transfer Conference, Brighton, 1994, 433-444.

[6] Cipollone E., Cumo M., Naviglio A.: *Condensation on lines of horizontal tubes for downflow of vapor at low velocity*, Atti 38 Congr. Naz. ATI, Bari, Vol. I(1983), 57-99.

[7] Davidson B. J., Rowe M.: *Simulation of power plant condenser performance by computational methods: an overview*, Power Condenser Heat Transfer technology, Hemisphere, Washington, 1981, 17-49.

[8] Donevski B., Nedelkovski I., Brodowicz K., Ostrowski K.: *Numerical simulation of flow and heat transfer in power plant condensers applying two-phase model*, Proc. of Int. Conf. on Fluid Engineering, Tokyo, Japan, Vol. 3(1997), 1707-1711.

[9] Honda H., Nozu S., Uchima B., Fujii T.: *Effect of vapor velocity on film condensation of R-113 on horizontal tubes in crossflow*, Int. J. heat Mass Transfer, Vol. 29(1986), 429-438.

[10] Marto P. J.: *Heat transfer and two-phase flow during shell-side condensation*, Heat Transfer Engineering, Vol. 5 (1984), No. 1-2, 31-61.

[11] Patankar V. S.: *Numerical heat transfer and fluid-flow*, Hemisphere, Washington 1980.

[12] Zhang C., Sousa M. C. A. and Venart S. E. J.: *Numerical simulation of different types of steam surface condensers*, Journal of Energy Resources Technology, Vol. 113 (1991), 63-70.

[13] Zhang C., Bokil A.: *A quasi-three-dimensional approach to simulate the two-phase fluid flow and heat transfer in condensers*, Int. J. Heat Mass Transfer, Vol. 40 (1997), No. 15, 3537-3546.

Symulacja numeryczna przepływu pary w skraplaczach z zastosowaniem obliczeń trójwymiarowych

Streszczenie

Praca ta zawiera wyniki symulacji numerycznej przepływu i kondensacji pary w eksperymentalnym skraplaczu, z zastosowaniem dwu- i trójwymiarowego matematycznego modelu obliczeniowego. Dla porównania zaprezentowano wyniki eksperymentalne, jak również, wyniki dwu- i trójwymiarowych obliczeń, opublikowane w wyszczególnionej literaturze. Na bazie przeprowadzonej analizy, można stwierdzić, że wyniki obliczeń z zastosowaniem trójwymiarowej symulacji numerycznej nie zawierają znaczących różnic w stosunku do dwu- i quasi- trójwymiarowych obliczeń, dla warunku brzegowego na wlocie do skraplacza w postaci stałego pola prędkości pary.

Należy jednak podkreślić, że zalety modelu trójwymiarowego są widoczne dla przypadku obliczeń, w których zastosowano niejednolite pole prędkości jako warunek brzegowy na wlocie do skraplacza. To niejednolite pole prędkości, odpowiada rzeczywistym warunkom panującym w skraplaczu, umiejscowionym na wylocie z turbiny. Otrzymane wyniki potwierdzają konieczność prowadzenia symulacji numerycznych przepływu pary zarówno w turbinie jak i w skraplaczu. Pole prędkości na wylocie z turbiny, otrzymane z symulacji numerycznej turbiny, będzie wykorzystane jako warunek brzegowy na wlocie do skraplacza, do dalszych obliczeń cieplno-przepływowych w przestrzeni skraplacza.