ISSN 1392 - 1207. MECHANIKA. 2006. Nr.5(61)

Coupled FEM simulation of turbulent flow and temperature in insulated pipes

A. Kačeniauskas*, J. Česnienė**

*Vilnius Gediminas Technical University, Saulėtekio 11, 10223 Vilnius, Lithuania, E-mail: arnka@fm.vtu.lt

**Lithuanian Energy Institute, Breslaujos 3, 44403 Kaunas, Lithuania, E-mail: cesniene@isag.lei.lt

1. Introduction

The design of district heating systems and satisfaction of consumer demands is related to advantageous heat supply and control technologies, where accurate evaluation of nonsteady thermo-hydraulic behaviour of heat carrying media is required. Non-steady thermo-hydraulic effects are associated with time lag between heat generation and consumption as well as with temporary storage of heat in a system. The turbulent flow in a pipe network poses a problem of a nonlinear nature. Moreover, in heat transfer analysis, convection and mass transport are strongly related to fluid flow problem. The physical properties of fluid, involved in both fluid flow and heat transfer analysis, depend on the temperature. In this case coupling of two problems arises and mentioned aspects require simultaneous modelling of turbulent flow and nonsteady heat transfer. In engineering practice, evaluation of different parameters, such as flow velocities, flow temperature, heat losses as well as turbulence factors, are simplified or even neglected for simplicity reasons.

Simplified engineering model of district heating network is usually presented as a grid composed by one-dimensional segments [1]. The oldest and perhaps the most widely used method for hydraulic analysis of pipe networks is H. Cross method [2]. However, when larger and more complex networks were analysed, this method was found to have convergence problems, and the application of the Newton-Raphson method [3] is recommended. In such works, the velocity distribution in a cross-section of a pipe is approximated by the averaged velocity. An accurate analysis of nonsteady flows [4] requires the evaluation of velocity distribution as well as turbulent effects [5]. Viscous time-dependent 2D or 3D flows are described by the Navier-Stokes equations. Thus, three or four equations should be solved numerically in order to obtain velocity distribution in the pipe. Moreover, evaluation of turbulent effects also requires enormous amount of information for complete description of the investigated flow.

The origin of turbulence modelling dates back to the end of the nineteenth century (1895), when Reynolds published his research on the time-averaged Navier-Stokes equations. Van Driest [6] devised a viscous damping correction for the mixing-length model, which is included in virtually all algebraic turbulence models in use today [7]. The one-equation turbulence models [8] have not enjoyed popularity and success in the community of engineers. Kolmogorov's k-ω turbulence model [9] was the first two-equation turbulence model. Lounder and Spalding's k-ε turbulence model [10] is the most widely used two-equation model. Even its demonstrable inadequacy for flows with adverse pressure gradient has done little to discourage its widespread use [11]. Sufficient computer resources become available to permit development of second-order closure models [12] based on computation of Reynolds-stress tensor. For a 3D flow such models introduce seven additional equations. In large eddy simulation (LES), the largest eddies are computed directly and the influence of the smallest eddies is modelled [13]. The method, which requires the most computer resources, is direct numerical simulations (DNS) that perform a complete time-dependent solution of the Navier-Stokes equations. At the most fundamental level [14] DNS can be used to obtain understanding of turbulence structure and processes, but hardly can be useful for engineering purposes.

Heat loss, the temperature distribution through the network and its value at consumer installation is an object for the estimation. Heat transfer problem from fluid to surrounding environment constitutes a multistep process: from a warmer fluid to a wall, through the multilayered wall, then to a colder environment. Heat transmission with mass transport takes place in fluid flow in a pipe, which is dominantly convective flow. The popular models for the estimation of heat losses in a certain location in the pipe examine the problem in two dimensions [15]. One class of approaches uses analytical expression [16] to estimate heat transfer, while another class of approaches uses a numerical integration of the governing differential equations [17]. For the simulation of heat losses along the pipelines a number of simplification is employed in order to save computational resources. In engineering approaches, radial heat transfer from heat carrying media to surroundings is assumed. However, in this estimation the influence of temperature drop in the network is neglected [18]. In practical application [19], the evaluation of thermal parameters, such as heat losses, is simplified by compounding the heat transfer coefficients into one overall coefficient, which should be estimated additionally. Numerical methods have been implemented in engineering programs specialized on modelling of district heating systems, such as TERMIS [19] and TINKLAS [20].

The finite element method (FEM) [21] has been frequently applied for heat transfer simulation in insulated pipes. The heat flow analysis in a pipe as an axi-symmetric cylinder has been presented in [22]. It was assumed that heat is flowing through a circular section of layered insulated material in one-dimension. The accuracy has been increased by subdividing the domain into a number of low-order finite elements [23]. The 2D problem of heat loss from a buried pipe has been presented in [24]. Coupled thermo-hydraulic analysis has been performed solving applications in different areas [25, 26]. It was observed, that the numerical modelling of heat losses along the pipelines requires 2D or 3D models that demand extensive computer processing capacity.

The dynamics of thermo-hydraulic phenomena affects the performance of district heating systems. It is very important to determine the time delay, which occurres distributing hot water through pipes. This delay is caused by heat capacity of the pipes as well as long network distances [27]. An accurate dynamic analysis requires exact evaluation of velocity distribution in pipes as well as that of turbulent effects. Physical fluid properties employed in corresponding flow analysis are temperature dependent. Thus, the evaluation of temperature dependent properties requires a simultaneous fluid flow and heat transfer analysis with the evaluation of coupling effects.

2. Mathematical model

Newtonian flow of viscous incompressible fluid is described by the Navier-Stokes equations [4]

$$\left(\frac{\partial\rho(T)u_i}{\partial t}+\frac{\partial\rho(T)u_ju_i}{\partial x_j}\right)=\rho(T)F_i-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu_e\frac{\partial u_i}{\partial x_j}\right) \quad (1)$$

$$\frac{1}{\gamma}\frac{\partial p}{\partial t}+\frac{\partial\rho(T)u_j}{\partial x_j}=0 \quad (2)$$

where u_i are velocity components; p is pressure; T is temperature; ρ is temperature dependent density; F_i are gravity force components; γ is bulk modulus equal 10^{15}. In Eq. (1), μ_e is effective viscosity

$$\mu_e=\mu(T)+\mu_t(k,\varepsilon,T) \quad (3)$$

here μ is the viscosity coefficient, which depends on temperature, and μ_t is turbulent viscosity.

$$\mu_t=C_\mu\rho(T)\frac{k^2}{\varepsilon} \quad (4)$$

here C_μ is turbulence model constant; k is turbulent kinetic energy; ε is turbulent kinetic energy dissipation rate. The solution of turbulence model then revolves around the solution of differential equations defining k and ε. The standard two equation k-ε model [10] is employed in this work, because it is widely applied for the solution of different engineering problems. The distribution of turbulent kinetic energy is described by the equation

$$\left(\frac{\partial\rho k}{\partial t}+\frac{\partial\rho u_jk}{\partial x_j}\right)=\frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right)+\mu_t\Phi-$$
$$-\rho\varepsilon+\frac{C_4\beta\mu_t}{\sigma_t}\left(F_i\frac{\partial T}{\partial x_i}\right) \quad (5)$$

here β is the coefficient of thermal expansion; σ_k is Schmidt number for the turbulent kinetic energy; σ_t is Prandtl number; Φ is viscous dissipation

$$\Phi=\mu\left(\frac{\partial u_k}{\partial x_j}+\frac{\partial u_j}{\partial x_k}\right)\frac{\partial u_k}{\partial x_j} \quad (6)$$

The distribution of turbulent kinetic energy dissipation rate is described by the equation

$$\left(\frac{\partial\rho\varepsilon}{\partial t}+\frac{\partial\rho u_j\varepsilon}{\partial x_j}\right)=\frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial x_j}\right)+C_{1\varepsilon}\mu_t\Phi\frac{\varepsilon}{k}-C_2\rho\frac{\varepsilon^2}{k}+$$
$$+\frac{C_\mu(1-C_3)\beta\rho k}{\sigma_t}\left(F_i\frac{\partial T}{\partial x_i}\right) \quad (7)$$

here σ_ε is Schmidt number for kinetic energy dissipation rate; $C_{1\varepsilon}$, C_2, C_3, C_4 are constants of the turbulence model. The final terms in Eqs. (5) and (7) describes the effects of buoyancy that are neglected in this work. Default values for the constants in the standard model are provided in [28]

$$C_{1\varepsilon}=1.44;\ C_2=1.92;\ C_3=1.0;\ C_4=0,$$
$$C_\mu=0.09;\ \sigma_k=1.0;\sigma_t=0.9;\ \sigma_\varepsilon=1.3;\ \beta=0 \quad (8)$$

The first law of thermodynamics and Fourier's law of heat conduction describe thermal fields. The non-linear parabolic equation of temperature conduction and convective mass transport is considered [23]

$$\left(\frac{\partial\rho(T)c(T)T}{\partial t}+u_j\frac{\partial\rho(T)c(T)T}{\partial x_j}\right)-\frac{\partial}{\partial x_j}\left(k_{ej}\frac{\partial T}{\partial x_j}\right)=\hat{q} \quad (9)$$

here $c(T)$ is specific heat, which might depend on temperature; $\hat{q}$ is heat generation rate per unit volume; k_{ej} are coefficients of effective conductivity that are influenced by the turbulence model and can be defined by the formula

$$k_{ej}=k_j(T)+\frac{\mu_t(k,\varepsilon,T)c(T)}{\sigma_t} \quad (10)$$

here $k_j(T)$ is temperature dependent conductivity. The Eq. (9) is nonlinear and requires iterative solution procedure. Non-linear effects caused by the temperature dependent material properties can be very significant when large variation of temperature occurs. Eqs. (1), (2), (5), (7), (9) presents strongly coupled mathematical model of the problems investigated in this work.

3. Problem description

The district heating network consists of long insulated pipes. Local losses and other local effects are small and can be neglected, therefore, turbulent flow and the temperature was investigated in a single insulated pipe. The geometry and boundary conditions of the problem are illustrated in Fig. 1. Due to axial symmetry of a pipe, only a half of it is considered in the 2D solution domain. Gravity force and buoyancy effects are neglected, because they have negligible influence on velocity distribution in a long pipe.

The solution domain and boundary conditions for the flow analysis are presented in Fig. 1, a. The velocity profile at the inflow is obtained from the experimental measurements [29]. Zero pressure boundary conditions are applied at the outflow. The full pressure variation could be evaluated at the post-processing step. No-slip velocity boundary conditions are prescribed on the pipe wall. The zero Dirichlet boundary conditions for the y component of

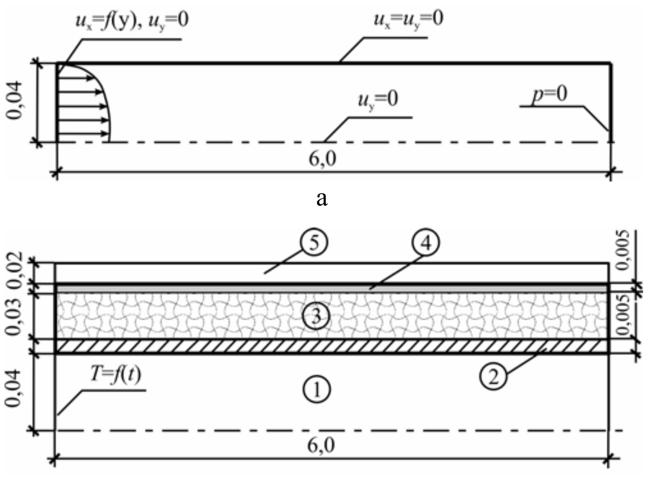

Fig. 1 Geometry and boundary conditions of the problem: a – solution domain for flow analysis; b – solution domain for coupled thermo-hydraulic analysis

velocity are specified on the rotating symmetry axis. The turbulence variables k and ε are defined at the inflow

$$k = \frac{3}{2}(0.01v)^2, \ \varepsilon = \frac{C_\mu k^{3/2}}{0.01L} \tag{11}$$

here L is characteristic length; $|v|$ is the magnitude of input velocity. The values of corresponding coefficients are equal 0.01. Smooth walls of the pipe are considered.

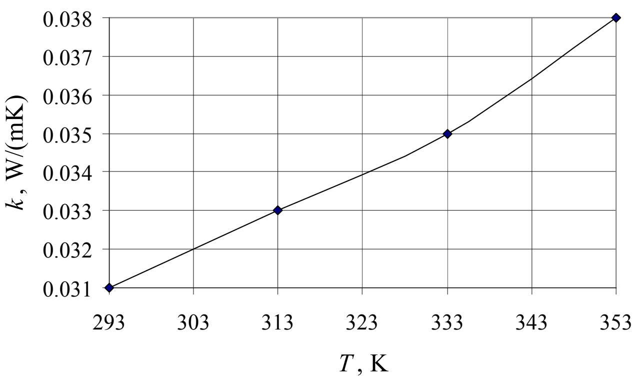

Fig. 2 Temperature dependent conductivity of polyurethane insulation

Performing coupled thermo-hydraulic analysis the solution domain should be extended and temperature distribution in insulation layers should be considered (Fig. 1, b). The extended solution domain consists of flow field *1*, steel pipe *2*, polyurethane insulation *3*, casing *4* made of high density polyethylene and air *5*. Material properties, that are independent on temperature in the investigated range of temperature values, are provided in Table 1. Temperature variation in air region is very small, therefore, material properties of the air are treated as being constant. Conductivity values of polyurethane (Fig. 2) are temperature dependent [30]. Values of temperature dependent water properties, such as density, dynamic viscosity and specific heat, are available in [29]. Initial temperature is equal 293 K in the whole solution domain. Time-dependent Dirichlet's boundary conditions for the temperature are prescribed at the inflow. Initially, the temperature of supplied water is equal 333 K. Later it linearly increases and at the end of time interval becomes equal 353 K. The standard Newman's boundary conditions are defined on the rest part of the boundary.

Table 1

Material properties

	ρ, m^3/kg	k, W/(mK)	c, J/(kgK)
Steel	7850	51.9	470
Polyurethane	86.5	$k(T)$	1470
Polyethylene	945	0.4	1900
Air	1.21	0.026	1000

4. Numerical results and discussions

The numerical analysis was performed by the commercial FEM software ANSYS [280]. The solution domain presented in Fig. 1, b was sub-divided into 15600 finite elements FLUID141 (16227 nodes). The structured finite element mesh was refined at the wall in order to perform accurate boundary layer analysis. The time interval [0, 300] s was sub-divided to 10000 time steps, because of limitations caused by the convection terms in Eqs. (1), (5), (7) and (9).

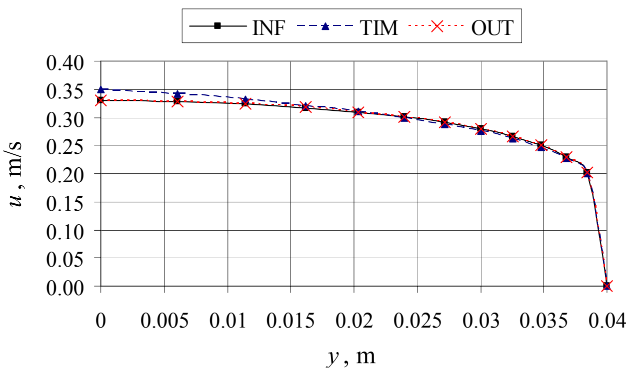

Fig. 3 Distribution of the velocity: boundary conditions at the inflow (INF); computed velocity profile at the outflow, when t=300s (OUT); computed velocity profile at the outflow, when t=9s (TIM)

The main attention investigating the results of flow analysis was focused on the velocity distribution, because it has essential influence on convective mass transport and final distribution of temperature. Fig. 3 illustrates velocity profiles at the outflow. At the beginning of the time interval (t=9 s) the velocity values at the outflow can be by 6% higher than that at the inflow. Later the developed turbulent flow becomes quasi-steady and velocity profile at the outflow becomes nearly equal to the velocity profile at the inflow. Fig. 4 illustrates the distribution of turbulent kinetic energy at the beginning of the pipe, when t=300s. At the beginning of the time interval, higher k values concentrates in the boundary layer near the wall. Later the area influenced by the turbulence grows and reaches the rotating symmetry axis. At 1-1.5m from the inflow the region of intensive mixing is formed. The values of effective viscosity as well as the distribution of the velocity are influenced by the turbulence effects in this region [29].

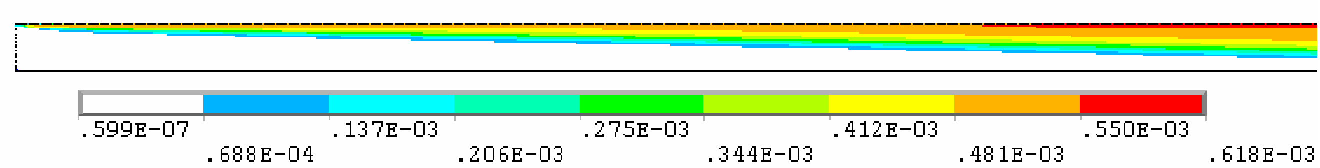

Fig. 4 Distribution of turbulent kinetic energy k in the 1m length pipe segment at the beginning of the pipe, when t=300 s

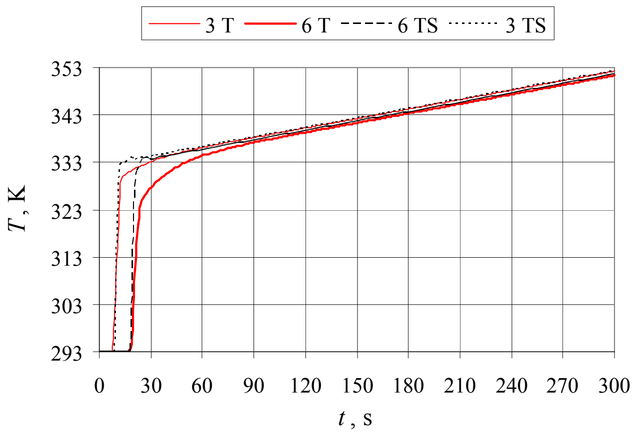

Fig. 5 Time variation of temperature in the points located at the rotating symmetry axis (3 – 3 m from the beginning of the pipe, 6 – 6 m from the beginning of the pipe. T – coupled thermo-hydraulic analysis, TS – separated thermal analysis)

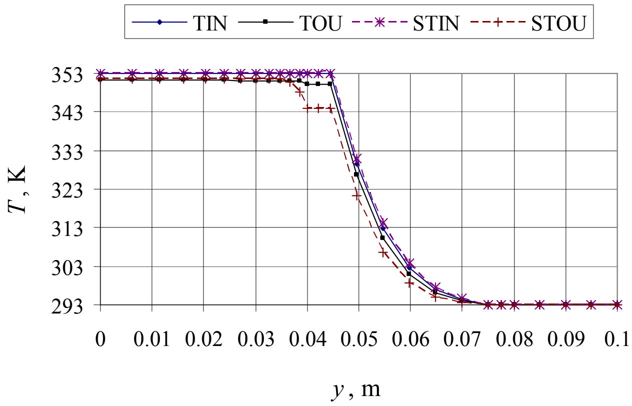

Fig. 6 Distribution of the temperature: TIN – initial cross-section, coupled thermo-hydraulic analysis; TOU – final cross-section, coupled thermo-hydraulic analysis; STIN – initial cross-section, separated thermal analysis; STOU – final cross-section, separated thermal analysis

In order to evaluate the accuracy of engineering methods temperature fields obtained by the coupled thermo-hydraulic analysis were investigated comparing them with the results of separated thermal analysis. The constant flow velocity u_x=0.3m/s corresponding flow rate created by the velocity profile (Fig. 3) was considered. The constant velocity was substituted into convective terms of Eq. (9) and separated thermal analysis was performed. The solution domain was sub-divided into 46800 finite elements (48267 nodes). The time interval [0, 300]s was subdivided to 30000 time steps equal 0.01s. The limitations caused by convection terms become very strict, because of the absence of turbulence increasing the effective conductivity in Eq. (10) and smoothing thermal field. Only discretisation parameters and stabilisation schemes [31] could be applied for damping of numerical oscillations.

Fig. 5 shows time histories of temperature evolution in several points distributed at the rotating symmetry axis. The results obtained by separated thermal analysis have oscillating character at the propagating thermal front. The results of coupled thermo-hydraulic analysis are significantly smoother and more reliable. At the propagating thermal front the values of temperature obtained by different analyses have significant differences, but later both numerical solutions become very similar. Thus, coupled analysis is necessary in case of dominant convective transport. Engineering methods [27] could cause inaccuracies observed computing time delay between heat generation and consumption. The biggest difference could be observed at the end of the pipe (at the cross-section located 6m from the inflow).

In Fig. 6 final distribution of the temperature is compared at the end of the pipe as well as that at the beginning. The results obtained at the inflow cross-section are almost identical, while at the end of the pipe numerical errors of separated analysis are accumulated. Thus, different values are obtained in steel pipe and a part of polyurethane insulation. The error of separated analysis is up to 6K at the final cross-section of the pipe. In the regions of propagating thermal front, where convective transport plays a dominant role, even larger differences could be obtained. The coupled thermo-hydraulic analysis takes long computing time. The longest numerical experiment continued 72 hours and 50 minutes on the standard PC (3 GHz Intel 4 processor, 512 RAM). Such computations are very useful for validation of simplified engineering models, but hardly can be extensively applied in practical design.

5. Conclusions

The coupled finite element analysis was applied to numerical investigation of turbulent flow and time-dependent temperature field in the insulated pipe. The coupled thermo-hydraulic analysis included turbulent viscous flow model, heat conduction and convective heat transfer model, temperature dependent material properties and multiphysical interactions.

The performed analysis of turbulent flow showed that turbulence effects have no significant influence on the velocity distribution at the outflow, when flow character becomes quasi-steady. Coupled thermo-hydraulic analysis is necessary for investigation of short time intervals and quickly changing flows with dominant convective transport. The numerical errors of separated thermal analysis assuming constant velocity of the flow accumulates at the end of the pipe. Coupled FEM analysis requires long computing time, therefore, parallel or distributed computing could serve as promising avenue for scientific investigation of district heating systems.

Acknowledgements

The work was supported by the Lithuanian Science and Studies Foundation under the contract No. T-05043. The authors would like to thank Dr. I. Gabrielaitienė for helpful suggestions and useful discussions on the district heating systems.

References

1. **Randlov, P.** District Heating Handbook. -European District Heating Pipe Manufacturers Association, 1997.-319p.
2. **Stephenson, D.** Pipeline Design for Water Engineers.-New York: Elsevier Scientific Publishing Company, 1981. -247 p.
3. **Zhao, H.** Analysis, Modelling and Operational Optimisation of District Heating Systems.-Lyngby: Technical University of Denmark, 1995.-226p.
4. **Baker, A.J.** Finite Element Computational Fluid Mechanics.-New-York: McGraw-Hill, 1983. -315p.
5. **Wilcox, D.C.** Turbulent Modeling for CFD.-La Cañada, California: DCW Industries, Inc., 1993. -400 p.
6. **Van Driest, E.R.** On Turbulent flow near a wall. -J. Aeronautical Sciences, 1956, No 18, p.1007-1018.
7. **Liu, C.H., Li, Y.** Turbulence modeling for computing viscous high-reynolds-number flows on unstructured meshes. -Computer Methods in Applied Mechanics and Engineering, 2001, No190(40-41), p.5325-5339.
8. **Spalart, P.R., Allmaras, S.R.** A One-Equation Turbulence Model for Aerodynamic Flows. -AIAA Paper 92-439, Reno, 1992.
9. **Kolmogorov, A.N.** Equations of turbulent motion of an incompressible fluid. -Izvestia Academy of Sciences USSR, Physics, 1942, No 6(1-2), p.56-58.
10. **Launder, B.E., Spalding, D.B.** The numerical computation of turbulent flows.-Computer Methods in Applied Mechanics and Engineering, 1974, No3, p.269-289.
11. **Leupi, C., Altinakar, M.S.** Finite element modelling of free-surface flows with non-hydrostatic pressure and *k-ε* turbulence model. -Int. J. Numerical Methods in Fluids, 2005, No 49(2), p.149-170.
12. **Burchard, H., Bolding, K.** Comparative analysis of four second-moment turbulence closure models for the oceanic mixed layer. -J. Physical Oceanography, 2001, No 31(8), p.1943-1968.
13. **Hanjalic, K.** Will RANS Survive LES? A view of perspectives. Transactions of the ASME. -J. Fluids Engineering, 2005, No 127(5), p.831-839.
14. **Sharma, N., Patanker, A.** A Fast computation technique for the direct numerical simulation of rigid particulate flows. -J. Computational Physics, No 205(2), 2005, p.439-457.
15. **Mei, V.C.** Heat transfer of buried pipe for heat pump application. -ASME J. Solar Energy, 1991, No113, p.249-250.
16. **Bau, H.H., Sdhal, S.S.** Heat losses from a fluid flowing in a buried pipe. -Int. J. for Heat Mass Transfer, 1982, No 25, p.1621-1629.
17. **Stevens, J.** Coupled conduction and intermittent convective heat transfer from a buried pipe. -Heat Transfer Engineering, 2002, No23(4), p.34-43.
18. **Engelmann, K., Krimmling, J.** Precalculating heat losses in district heating networks. -Euroheat and Power, 1998, No11, p.16-21.
19. TERMIS. Manual Version 1.2. -Seven Technologies A/S, 2001.-200p.
20. **Minkštimas, R., Masaitis, S.** District heating modelling software TINKLAS. -Proc. of Conf. Lithuanian Science and Industry: Heating Energy and Technology".-Kaunas, 2002, p.169-172 (in Lithuanian).
21. **Zienkiewicz, O.C., Taylor, R.L.** The Finite Element Method. v.1-3, the fifth edition.-London: Butterworth Heinemann, 2000.-712p.
22. **Rao, S.S.** The Finite Element Method in Engineering. -New York: Pergamon Press, 1982.-641p.
23. **Lewis, R.W., Morgan, K., Thomas, H.R., Seetharamu, K.N.** The Finite Element Method in Heat Transfer Analysis. -Wiley &.Sons Ltd., 1996. -274p.
24. **Schneider, G.E.** An Investigation into heat loss characteristics of buried pipes. -J. of Heat Transfer, Transactions of ASME, 1985, No 107, p. 696-699.
25. **Papadrakakis, M., Oñate, E., Schrefler, B.** (editors). Computational Methods for Coupled Problems in Science and Engineering. Barcelona, -CIMNE, 2005. -815p.
26. **Dim, J.R., Sakura, Y., Fukami, H.** Heat transport analysis and three-dimensional thermo-hydraulic simulation in the Ishikari basin, Hokkaido, Japan. -Hydrological Processes, 2002, No16(11), p.2221-2238.
27. **Gabrielaitienė, I.** The Modelling of District Heating Networks by the Finite Element Method. Doctoral Dissertation.-Vilnius Gediminas Technical University, 2003.-103p.
28. ANSYS Theory Reference, 8th edition, SAS IP INC., 1997.
29. **Kačianauskas, R., Kačeniauskas, A., Česnienė, J., Brinkienė, K., Kėželis, R., Juozapavičius, A.** Investigation of unsteady thermo-hydraulic processes in district heating system. Report T-05043, -Lithuanian Science and Studies Foundation, 2005.-56p.
30. LST ISO 8894-1:1996 Fire Resistant Materials. Determination of Thermal Conductivity. Part 1. Hot Wire Method (in Lithuanian).
31. **Kačeniauskas, A.** Parallel solution of convective transport equation by GLS stabilised space - time FEM. -Mechanika, 2003, No4(42), p.61-66.

A. Kačeniauskas, J. Česnienė

TURBULENCINĖS TĖKMĖS IR TEMPERATŪROS IZOLIUOTUOSE VAMZDŽIUOSE SUSIETOJI ANALIZĖ BAIGTINIŲ ELEMENTŲ METODU

R e z i u m ė

Straipsnyje pateikiamas skaitinis izoliuoto vamzdžio modelis, įvertinantis turbulencinį šilumnešio tekėjimo režimą bei greičio ir temperatūros laukų sąveiką, kai temperatūros laukai yra nenusistovėję. Darbe baigtinių elementų metodu išspręstas su 2D ašimi simetrinis izoliuoto vamzdžio uždavinys. Nenusistovėjusiems termohidrauliniams procesams izoliuotame vamzdyje modeliuoti atliktos dvi skirtingos skaitinės analizės.

Susietoji termo-hidraulinė analizė apima klampaus nespūdaus skysčio tėkmę, turbulencinį k-ε modelį, temperatūros laukus, medžiagų savybių priklausomybes nuo temperatūros bei skirtingų fizinių laukų tarpusavio sąveiką. Įvairiuose taškuose pateiktos temperatūros kitimo laikui bėgant kreivės bei temperatūros pasiskirstymas būdinguose uždavinio apibrėžimo srities pjūviuose. Įvertinti paklaidoms, kurias daro plačiai taikomi hidrauliniai modeliai, buvo atlikta atskirtoji šiluminė analizė. Atliktas kiekybinis skirtingų analizių rezultatų palyginimas.

A. Kačeniauskas, J. Česnienė

COUPLED FEM SIMULATION OF TURBULENT FLOW AND TEMPERATURE IN INSULATED PIPES

S u m m a r y

In this paper, the numerical model of thermally isolated pipe was designed including turbulence and flow velocity strongly coupled with temperature field. 2D axis-symmetrical problem of thermally insulated pipe was solved by finite element method. Two different numerical analyses were performed in order to simulate nonsteady thermo-hydraulic processes in the thermally insulated pipe.

The coupled thermo-hydraulic analysis includes viscous incompressible flow, k-ε turbulence model, heat conduction and convective heat transfer model, temperature dependent material properties and multi-physical interactions. Time histories of temperature evolution were provided in several characteristic points of the solution domain. Temperature distribution was examined at several cross-sections. In order to evaluate the characteristic errors of engineering hydraulic models, separated thermal analysis was performed independently. Quantitative comparison of the results obtained by the different numerical analyses was performed.

А. Каченяускас, Ю. Чеснене

СВЯЗАННОЕ МОДЕЛИРОВАНИЕ ТУРБУЛЕНТНОГО ПОТОКА И ТЕМПЕРАТУРЫ В ИЗОЛИРОВАННЫХ ТРУБАХ ПРИ ПОМОЩИ МЕТОДА КОНЕЧНЫХ ЭЛЕМЕНТОВ

Р е з ю м е

В статье представлена численная модель теплообмена в изолированной трубе, которая учитывает турбулентность потока и переменную скорость теплоносителя, сильно связанные с температурным полем. Двухмерная осесимметричная задача изолированной трубы решена при помощи метода конечных элементов. Два разных численных анализа произведены с целью моделирования нестационарных термогидравлических процессов.

Связанный термогидравлический анализ включает вязкую несжимаемую жидкость, κ-ε модель турбулентности, теплопроводность и конвективный транспорт, свойства материалов, зависящие от температуры и взаимодействие разных физических полей. В нескольких точках и характерных сечениях исследуются изменения температуры по времени. С целью определения погрешностей, допускаемых популярными инженерными методами, был произведён отдельный теплоанализ. Проведено количественное сравнение результатов, полученных при помощи разных методов анализа.

Received January 5, 2006