UDC 536.248.1

Romanuyk D. A., Panfilov S. V., Gromov D. S.

Solution of the conjugate problem of gas dynamics and heat transfer in structures with a large ratio of geometric scale values

Within the scope of the research work, we have developed the methods and software package for solving the conjugate heat and hydraulic problems based on the classical approach to performing hydraulic calculations and modeling thermal processes by means of the finite volume method in the ANSYS Fluent software package. The developed means allowed us to efficiently calculate the thermal state of complex technical objects. The study gives mathematical formulation of the methods and suggests the results of their approbation and verification.

Keywords: heat and mass transfer, computational simulation, conjugate problems.

Introduction

Computer-aided simulation technologies are taking the lead in development of various instruments, devices and structures. There are numerous software packages able to compute heat and mass transfer and gas dynamics processes as individual or conjugate problems. We should lay emphasis on fast-paced development of computer-aided simulation regarding the development of advanced source codes and enhancement of computational resources.

Meanwhile, there are some objects for which it is difficult to implement direct simulation of thermal and gas-dynamic processes using advanced CAE packages due to a variety of basic geometrical parameters. For example, considering a structure with overall dimensions of 20...30 mm with an available developed network of pipelines of 60…150 mm in diameter, the solution to conjugate problems of heat transfer and gas dynamics by using only *CFD* codes requires development of finite-volume grids consisting of tens and even thousands of millions of cells. This represents a practically unsolvable problem even via contemporary computing clusters. Ad hoc problem-solving approaches need to be found.

The staff of the Laboratory of Gas Dynamics and Heat and Mass Transfer at Design Bureau for Special Mechanical Engineering (JSC “KBSM”) was assigned to deal with the problem in question. To solve the problem, they proposed a conjugate thermohydraulic computation method and developed a software suite based on the application of the classic approach to hydraulic computations and on simulation of thermal processes using the finite-volume method in the *ANSYS Fluent* software environment.

Method description

Below we will consider the problem related to the thermal state of a concrete structure being heated and cooled by the air flowing through a complex pipeline system. As a result of heat exchange with concrete, the air temperature significantly changes as the air flows from the inlet point to the outlet point. To solve such a problem of heat transfer, we shall set boundary conditions on inner surfaces of pipes with account for the air temperature non-uniformity in pipes. Hydraulic characteristics of a system also depend on air temperature distribution.

Solving such a problem in unstable conditions by using only computational fluid dynamics methods is infeasible, since among other factors, it involves small time intervals to be applied. Based on the above, we shall use a software package intended for solving the heat transfer problem in the finite volume-based layout, as well as hydraulic computations in the classic layout.

In order to solve the heat transfer problem and as the basis for development, we use the *ANSYS Fluent* software package modified in terms of integration with a number of proprietary algorithms needed to take into account the specifics of heat and mass transfer processes in the structure in different operation modes.

Hydraulic computation of pipeline systems is performed using the *Hydra*, an in-house development by JSC "KBSM", which can simultaneously operate together with *ANSYS Fluent* in a continuous data exchange mode. The *ANSYS Fluent* and *Hydra* data exchange interface is implemented using a set of user-defined *UDF* functions.

Initial data for the Hydra software is the total air flow rate in the system (air volume or air mass flow rate) and the system inlet air temperature, as well as temperature distribution in the surrounding structure. The air low rate and temperature can be set as time-dependent functions. To set outside temperatures, we use a file for recording average temperatures of the surrounding concrete structure for each system section at each step of the problem-solving process while running the *ANSYS Fluent* package.

The *Hydra* software generates output data such as average values of air temperature and heat transfer coefficient for each section, recorded in a file used by the *ANSYS Fluent* package for setting boundary conditions on the pipe surface. During computations, a continuous data exchange is maintained between software packages.

This method enables the required recording of the following factors and processes:

- airflow in a pipe system with the relevant variable heat transfer into and out of a concrete structure;
- variation of thermal and physical properties of a structure (thermal conductivity and thermal capacity of materials) during the heating process;
- random setting of the law of air temperature variation at the system's inlet.

The *Hydra* software determines hydraulic characteristics and heat transfer parameters in accordance with the known dependencies derived from [1–3].

Mathematical model

The air mass flow rate in system pipelines is determined by using the mesh-current method based on the Kirchhoff laws. A certain number (N) of independent circuits are selected within the computational model of the pipeline system. The following condition shall be satisfied for each loop:

$$\sum_{j=1}^{M_i} \Delta P_j = 0, \quad i = 1, \ldots, N,$$

$$\Delta P_j = \left(\zeta_{_{Mj}} + \lambda_j \frac{L_j}{d_j} \right) \frac{G_j \left| G_j \right|}{2\rho_j F_j^2},$$

$$\lambda_j = \lambda_j \left(\mathrm{Re}_j \right), \ \ \mathrm{Re}_j = \frac{\left| G_j \right| d_j}{\mu_j F_j},$$

$$\rho_j = \frac{P_j}{RT_{sj}}, \ \ \mu_j = 1{,}71 \cdot 10^{-5} \frac{390}{T_{sj} + 117} \left(\frac{T_{sj}}{273} \right)^{3/2},$$

where M_i – number of sections in the *i*-th circuit;

ΔP_j – stagnation pressure reduction in the *j*-th circuit section;

$\zeta_{_{Mj}}$ – aggregate local drag coefficient in the *j*-th circuit section;

λ_j – friction drag coefficient in the *j*-th circuit section;

L_j – length of the *j*-th circuit section;

d_j – hydraulic diameter of the *j*-th circuit section;

G_j – air mass flow rate in the *j*-th circuit section ($G_j > 0$, if the flow direction coincides with the circuit bypass direction, otherwise – $G_j < 0$);

ρ_j – air density at temperature T_{sj};

F_j – passage area of the *j*-th circuit section;

μ_j – dynamic air viscosity at temperature T_{sj};

P_j – static pressure;

R – gas constant (for air $R = 287.7$ J/(kg·K));

T_{sj} – average air temperature in the *j*-th section.

The resulted system of non-linear equations is closed by balance ratios for flow rate values in nodes

$$\sum_{j=1}^{N_k} G_j = 0,$$

where N_k – number of sections converged in the *k*-th node;

| ISSN 2542-0542 Journal of "Almaz – Antey" Air and Space Defence Corporation | No. 3, 2017

$G_j > 0$, if the flow is direction toward the node, otherwise – $G_j < 0$.

This system of non-linear equations is solved iteratively, using one of the standard methods. As a result, parameters of all sections of a hydraulic system are determined.

Air inlet temperature T_1 is determined at each iteration of the solution for each section. Assuming that surrounding structure temperature T_e is constant along the section length, we can determine air inlet temperature T_2, average air temperature T_s in the section, and average coefficient of heat transfer α_s, using the following equations:

$$\Delta T = (T_e - T_1)\left[1 - \exp\left(\frac{\alpha_s S}{C_p G}\right)\right],$$

$$T_2 = T_1 + \Delta T,$$

$$T_9 = T_e - \Delta T \frac{C_p G}{\alpha_s S},$$

$$\alpha_s = 0.023 \frac{\lambda}{d} \mathrm{Re}_d^{0.8} \mathrm{Pr}^{0.4}.$$

Here, S – heat exchange surface;

C_p – isobaric air thermal capacity;

G – air mass flow rate;

λ – air thermal conductivity coefficient;

d – hydraulic diameter of pipe;

Re_d – Reynolds number calculated by hydraulic diameter;

Pr – Prandtl number (for air, Pr = 0.7).

Temperature T_e for each section is determined from the solution to the heat transfer problem via the *ANSYS Fluent* package, while average air temperature T_s and average coefficient of heat transfer α_s are sent to *ANSYS Fluent* as the boundary conditions at each computation step.

Testing and verification

The computation method and software tools were tested and verified based on experimental studies intended to investigate thermal conditions of a structure fragment. This fragment includes a steel structure, heating pipelines, a concrete filler, a reinforced central filler, and heat insulation. Its geometrical shape is a segment with a plane angle of 45° and height of ~5 m.

The experiment intended to heat up the fragment was conducted for 120 days in accordance with the heating cyclogram, including non-stop recording of data from thermal sensors mounted at the key points of the structure.

The developed 3D geometrical model comprises all basic structural elements of a test bench. Assumptions and approximations made during development of the computational model, as well as boundary conditions, properties of materials and heating cyclogram were taken based on their best possible compliance with the experiment conditions.

Comparison was conducted based on measurement data taken at 30 points located on seven measuring levels. As an example, the figure shows computation results compared with sensor readings on one of the measuring levels.

Comparison of curves of time-dependent temperature variations at reference structural points demonstrates the qualitative resemblance of computation results and experimental data. Comparison of temperatures measured for the 120-day period (steady-state thermal condition or near condition) shows divergence of 12 % maximum between temperature computation results and experimental data. Generally, computation results are in good agreement with experimental data, thus proving that the method and the developed software tool are considered fit for calculations of such structures.

Moreover, we compared the results of the solution to the "pipe in concrete" test case using the developed method and direct solution of the conjugate problem via the *ANSYS Fluent* software package.

We studied the problem of the steady-state hot airflow in a pipe of 68 mm in diameter located inside a cylindrical concrete structure 500 mm in diameter and 5 m in length. The preset air flow velocity at the pipe inlet is equal to 8 m/s, the air

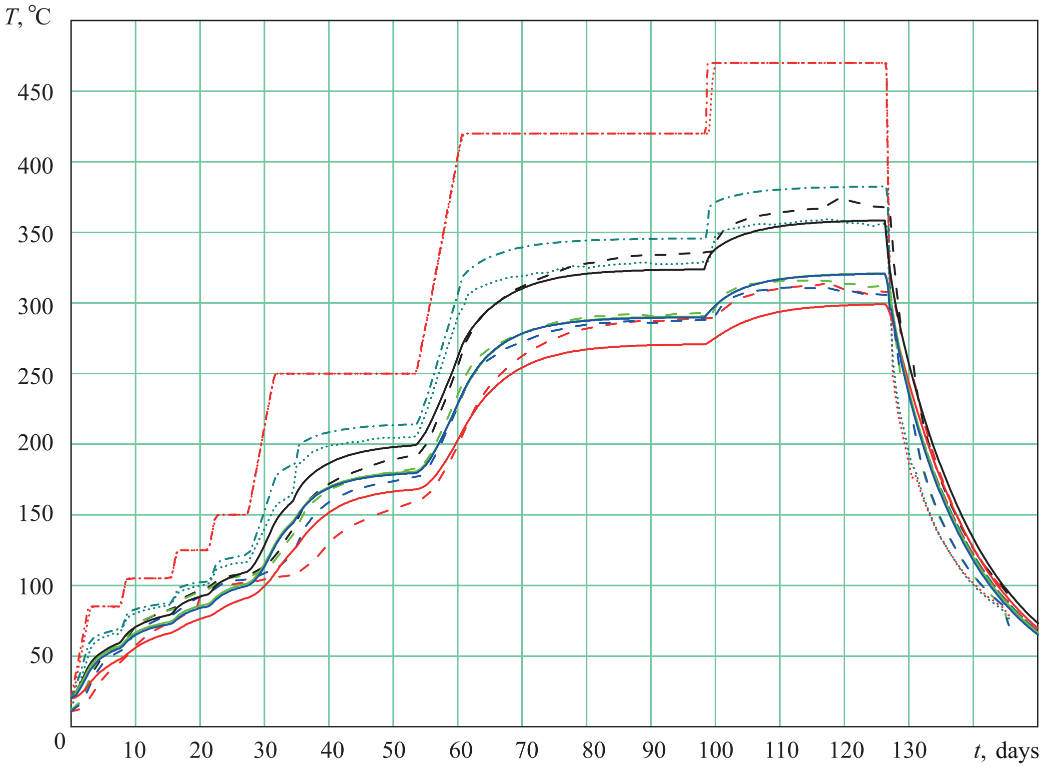

Figure. Comparison of computation results with sensor readings:
·····, – · · – T1 (exp., computation); ·····, – · · – T3 (exp., computation); — –, —— – TB-10.7 (exp., computation);
– –, —— – TB-10.8 (exp., computation); — –, —— – TM-10.11 (exp., computation);
– –, —— – TM-10.12 (exp., computation)

temperature is 480 °C. The convection boundary condition is applied to the external boundary of a concrete structure.

The computation domain was divided into tetrahedral cells, while prismatic cells near the wall inside the pipe were used for resolving the boundary layer problem. For computations, we used grids with cells different in sizes, using several turbulence models.

The table contains the values of specific rate of heat flow (W/m^2) from the hot air to a concrete structure, which are calculated using different methods. Here, the following designations are introduced: *Hydra* – solution using the developed method; *NS* – Navier – Stokes equations (without turbulence model); *SA* – Spalart – Allmaras model; *SST* – Menter $k-\omega$ *SST* model; *KE* – $k-\varepsilon$ model; *KE_ew* model $k-\varepsilon$ with improved near-wall functions; *KE_av* – average value between two $k-\varepsilon$ models; Δ – deviation of computation result from the *Hydra* based solution.

The analysis of table data shows that:

Results of specific heat flow rate computations

Cell size, mm	Prismatic layer, mm	Number of cells, thous.	*Hydra*	*NS*	*SA*	*SST*	*KE*	*KE_ew*	*KE_av*
10	–	900	2827	1430	2477	2537	2928	2690	2809
5	0.5	3900	2829	1405	2402	2318	3124	2611	2868
3	0.3	8200	2825	1571	2312	2342	3192	2618	2905
2	0.2	18,200	2825	1811	2270	2365	3198	2652	2925
Deviation of computation results from the *Hydra software-based solution*									
Δ, %					–12.3	–10.2	3.6	–4.8	–0.6

• using of the Navier – Stokes equations gives knowingly reduced heat flows as the flow under consideration is the turbulence flow;

• all the turbulence models considered give a deviation from the basic variant (*Hydra*) within (–12.3…+3.6) %;

• the Spalart – Allmaras model is the most primitive one of all the models considered and gives the maximum deviation;

• comparison of results for other models shows a relatively small deviation;

• integral dependencies implemented in the *Hydra* computation model are recommended for thermohydraulic computations and give an error within 3…5 %.

Based on the above, we may conclude that the *Hydra* software allows to compute the heat transfer rate with the error of 3…5 % max.

Conclusion

The developed method allows to carry out computational simulation of thermohydraulic processes in large-sized structures, which are cooled or heated with the help of a pipe system with a gaseous or liquid coolant.

The method is intended for steady-state or transient process computations to determine structures’ thermal conditions, as well as for determining temperature fields in different operation modes with account for the setting of the conjugate problem heat transfer and gas dynamics in case the direct solution to the problem using the existing *CFD* codes is infeasible.

This method has the following significant advantages over direct simulation: the use of a considerably small amount of computational resources; a considerably higher computation speed without loss of accuracy.

The method is verified based on experimental data and by comparing with direct *CFD* computation methods. In order to certify the method and software to be used for projects related to Rosatom State Atomic Energy Corporation, the verification report and software certificate were issued.

Using the method described above, we successfully calculated various steady-state and transient thermal conditions of a concrete-steel structure of 28 m in diameter and of 20 m in height, tightly packed with equipment and a well-developed network of air purging pipelines 60…150 mm in diameter.

Bibliography

1. *Kutateladze S. S.* Teploperedacha i gidrodinamicheskoe soprotivlenie. M.: Energoatomizdat, 1990. 367 s. (Russian)

2. *Kreith F., Black W.* Basic Heat Transfer / Transl. from Eng. M.: Mir, 1983. 512 p.

3. *Idelchik I. E.* Spravochnik po gidravlicheskim soprotivleniyam / Pod red. *M. O. Shteinberga*. 3-e izd., pererab. i dop. M.: Mashinostroenie, 1992. 672 s. (Russian)

Submitted on 30.11.2017

Romanyuk Denis Andreevich – Deputy Head of Laboratory of Gas Dynamics and Heat and Mass Transfer, Joint Stock Company Design Bureau for Special Mechanical Engineering, Saint Petersburg.
Science research interests: gas- and hydrodynamics, heat and mass transfer processes, gas-particle flows, computational methods.

Panfilov Sergey Vladimirovich – Candidate of Physical and Mathematical Sciences, Chief Specialist of Laboratory of Gas Dynamics and Heat and Mass Transfer, Joint Stock Company Design Bureau for Special Mechanical Engineering, Saint Petersburg.
Science research interests: gas dynamics of technical systems, gas-particle flows, aerodynamics, heat exchange, computational methods.

Gromov Dmitriy Sergeevich – Candidate of Engineering Sciences, Head of Laboratory of Gas Dynamics and Heat and Mass Transfer, Joint Stock Company Design Bureau for Special Mechanical Engineering, Saint Petersburg.
Science research interests: heat exchange, thermal stabilization, heat transfer intensification, computational methods.

Решение сопряженной задачи газовой динамики и теплообмена в конструкциях с большим соотношением геометрических масштабов

В рамках выполнения научно-исследовательских работ созданы методика и программный комплекс для решения сопряженных теплогидравлических задач, основанные на использовании классического подхода к выполнению гидравлических расчетов и моделировании тепловых процессов с помощью конечно-объемного метода в программном пакете *ANSYS Fluent*. Разработанные средства позволяют эффективно рассчитывать тепловое состояние сложных технических объектов. Приведены математическое описание методики, результаты ее апробации и верификации.

Ключевые слова: тепломассообмен, численное моделирование, сопряженные задачи.

Романюк Денис Андреевич – заместитель начальника лаборатории газодинамики и тепломассообмена Акционерного общества «Конструкторское бюро специального машиностроения», г. Санкт-Петербург.
Область научных интересов: газо- и гидродинамика, процессы тепло- и массообмена, течения газовзвесей, вычислительные методы.

Панфилов Сергей Владимирович – кандидат физико-математических наук, главный специалист лаборатории газодинамики и тепломассообмена Акционерного общества «Конструкторское бюро специального машиностроения», г. Санкт-Петербург.
Область научных интересов: газовая динамика технических систем, течения газовзвесей, аэродинамика, теплообмен, вычислительные методы.

Громов Дмитрий Сергеевич – кандидат технических наук, начальник лаборатории газодинамики и тепломассообмена Акционерного общества «Конструкторское бюро специального машиностроения», г. Санкт-Петербург.
Область научных интересов: теплообмен, термостабилизация, интенсификация теплоотдачи, вычислительные методы.