Hybrid methods for simulating hydrodynamics and heat transfer in multiscale (1D-3D) models

S A Filimonov[1], E I Mikhienkova[2], A A Dekterev[1,2], D V Boykov[1]

[1] Kutateladze Institute of Thermophysics SB RAS, Lavrentevs 1630090, Novosibirsk, Russia
[2] Siberian Federal University, Svobodniy 79/10, 660041, Krasnoyarsk, Russia

E-mail: dekterev@mail.ru

Abstract. The work is devoted to application of different-scale models in the simulation of hydrodynamics and heat transfer of large and/or complex systems, which can be considered as a combination of extended and “compact” elements. The model consisting of simultaneously existing three-dimensional and network (one-dimensional) elements is called multiscale. The paper examines the relevance of building such models and considers three main options for their implementation: the spatial and the network parts of the model are calculated separately; spatial and network parts are calculated simultaneously (hydraulically unified model); network elements "penetrate" the spatial part and are connected through the integral characteristics at the tube/channel walls (hydraulically disconnected model). Each proposed method is analyzed in terms of advantages and disadvantages. The paper presents a number of practical examples demonstrating the application of multiscale models.

1. Introduction

In the contemporary world, the numerical simulation is practically indispensable when studying natural systems and designing technical objects. At solving problems of hydrodynamics, different modeling approaches are used depending on the type of the object (system, device, etc.). If the object can be represented in the form of a set of extended elements, in which the ratio of the characteristic dimensions differs in 10-1000 times, such as piping, channels, ducts, etc., than the modeling of such objects typically is carried out based on network (hydraulic) simulation [1, 2]. At that, the calculation of the desired characteristics of such a system (pressure, velocity, temperature, etc.) is carried out by methods based on the theory of hydraulic circuits (THC) [1]. The hydraulic circuit consists of a set of nodes and edges (a simple directed graph [3]). The laws of mass and energy conservation (Kirchhoff's laws) are ensured in the nodes, while the edges provide for the law of momentum, which describes the dependence of the pressure drop along the length of the edges depending on the flow rate.

Use of the computational fluid dynamics (CFD) methods allows obtaining the spatial distribution of flow characteristics and heat transfer in “compact” objects, that is the objects, whose ratio of the characteristic dimensions differs in 1-10 times [4, 5]. These methods are based on the numerical solution of the Navier-Stokes equations. As a consequence, we obtain fields describing concerned object characteristics distributed over the volume, for example: velocity, pressure, temperature, concentration, turbulence characteristics, etc.

Each of these simulation approaches has its advantages and disadvantages. The undoubted advantage of network simulation is the relative simplicity of constructing a topology of a model and a low computational cost. The disadvantages may include the strong dependence of the model on empirical data (e.g., hydraulic resistance), characterizing the properties of the element (node or edge) as well as the possibility of obtaining only system characteristics averaged over the network element. Strong point of spatial CFD simulation is the possibility of modeling an object with complex geometry and describing the detailed flow structure, as well as less dependence of the model on empirical data, however, such simulation requires significant computational costs. Both simulation approaches have found wide application in solving various practical problems. Nevertheless, there are cases where simulation of the studied object within a single approach is quite difficult. In this case it is necessary to build multiscale model, which allows describing objects consisting of "compact" and extended elements. Examples of such simulation objects can be found in various areas of life:

- blood-vascular system consisting of cardiac muscle, aorta, major arteries and veins, on the one hand, and networks of fine blood vessels and capillaries, on the other hand;
- thermal and water supply system containing long pipeline sections and complex regulating, distributing and collecting devices;
- natural waterway systems, which include river channels with canals and delta, as well as hydro-technical utilities.

2. Basic methods of constructing multiscale models

There are several ways of implementing multiscale models, among which the main are the follows:

1. *fully-separated model*: the spatial and the network parts of the task are calculated separately;
2. *hydradynamically unified model*: the spatial and the network parts are calculated simultaneously;
3. *hydradynamically disconnected model*: network elements "penetrate" the spatial part, and the connection is implemented by using the integral characteristics at the walls of the tube/channel.

2.1 Fully separated model

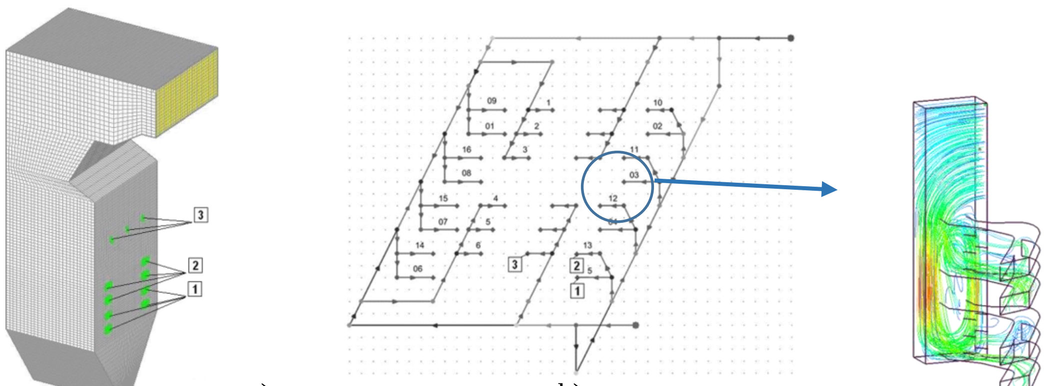
When applying this model, spatial and network parts of the task are calculated separately and usually by means of different software products, at that, the calculation data obtained from one of the model parts determine the input parameters of the another part. The most striking example of the application of this method is the determination of the hydraulic resistance of non-standard element of the network model by means of spatial methods [11], or the definition of the input parameters of a complex device, after having calculated the lead-in pipeline fittings by network methods. Undoubted advantage of this method is the possibility to use ready-made software products to calculate both parts of the task without any modifications, because the transmission of information between the parts of the model is carried out through the peripheral data transfer. Though use of this modeling technique requires fulfillment of one of the following conditions that severely restricts its applicability:

- for the element, which is simulated by a three-dimensional calculations, there must be opportunity to build the relationship between flow rates in various inlets of the spatial element and pressure drops between them that may require a very large number of 3D calculations;
- pressure loss in the 3D element must have a negligibly small network effects on the network at large.

However, if one of the conditions is satisfied, then for further wide-sweeping calculations we can use only the network model, and hence reduce the total time required for the computational research.

Let demonstrate the application of the completely separated simulation method on the example of calculation of the exhaust ducts of the hot air supply system for the boiler at Yavinsky state district power plant (GRES). To determine the boundary conditions of the spatial model of the combustion chamber (Fig.1a) we have constructed a network model of the hot air supply system (Fig. 1b). The comparison of calculation results of the network model with the experimentally measured data on the hot air flow rate revealed a large discrepancy between the experimental data and calculation. Analysis of the results has shown that the problem lies in the duct, distributing out the hot air to the burner tiers (Fig. 1c). It should be noted that the duct design has one inlet and four outlets that in general case would require a large number of 3D calculations to determine the parameters of the network model. However, in this task the pressures at all four duct outlets are similar (equal to the pressure in the combustion chamber), and thus all the settings required for the network calculation can be obtained from a single 3D calculation.

The analysis of the spatial calculation of the duct revealed the presence of a large vortex in the delivery duct, which locks the flow and causes the non-uniformity of flow rates at the four outlets. Not considering the effect of resistance generated by the vortex leads to the discrepancy between the experimental data and results of the network simulation.



a) b) c)

Fig. 1. The model of the hot air supply system for the boiler at Yavinsky GRES.
a) spatial model of the combustion semi-chamber of the boiler;
b) a network model of the hot air supply system;
c) delivery duct of the secondary blast; 1, 2, 3– burner tiers

2.2 The hydrodynamically inified model

If building a separated model for a certain task is impossible, i.e. it is impossible to obtain a definite dependence of the parameters of one part of the model on the calculation result of the other part, than it is necessary to use a hydrodynamically unified multi-scale model. In this approach, both model parts are calculated simultaneously, and the data exchange between the parts occurs iteratively. In the vast majority of applications of this method, different parts of the model are calculated by means of different software products. Therefore, there is a need to transfer data from one program to another using either third-party libraries, for example, MpCCi library is used to link ANSYS and Flowmaster [7], or via additional exchange files [8]. Even if the software package allows simulating both model parts, even then the problem of boundary conditions transfer remains relevant. For example, in [9] dealing with the simulation of blood flow, the values of flow rate are transferred downstream, while pressure values are transferred upstream.

The SigmaFlow software package formulates a single equation for the pressure correction for the entire computational domain, including the network and the spatial parts. This allows calculating the associated pressure field, while flow rates at the boundaries are calculated by interpolation (see [10]). This approach provides a faster convergence compared to traditional methods, as well as leads to unambiguous determination of the flow direction.

The example of practical use of such approach is the problem on simulation of the gas removal system at the aluminum plant facility [11]. The gas removal system is designed to collect from electrolysis baths harmful gases and transfer them to the gas cleaning devices. The problem consists in the equalizing of flow rates from each bath in four spreads (1st and 2nd spreads contain 24 baths, 3rd spread contains 26 baths, and 4th spread contains 20 bath). The equalizing of the gas flow rates removed from the baths is achieved by means of the regulating gate valves. The main feature of the gas removal system consists in presence of the central collecting receiver, where pressure losses reach half of the total pressure loss in the investigated part of the system. In the general case, the simulation of this problem within the framework of one of the main approaches (fully 3D or fully HCT) is impossible. The size of the computational grid to build a fully three-dimensional model of the whole facility will be too large, since the lengths of certain linear sections are more than 100 m. When building a fully-netted model it is necessary to know the hydraulic resistance of the collecting drain for different operating modes. The use of separated model with the definition of hydraulic resistance of the collecting drain and each branch tube based on preliminary spatial calculations is also not suitable, since the flow distribution along the ducts varies significantly in the regimes before and after flow rates equalization. Building a multi-scale model allows removing this obstacle. Extended elements are arranged in the form of a network, a collecting drain is presented in the form of spatial element (see Fig. 2), and the resistance of the collecting drain is calculated for specific flow rates.

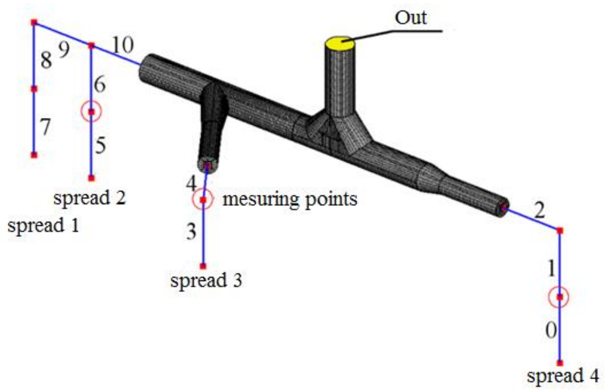


Fig. 2 Multi-scale model of the gas removal system at the facility of aluminum plant. No. 1-4 – are the numbers of the bregades, 0-10 – are the numbers of calculation edges, circles show measuring points.

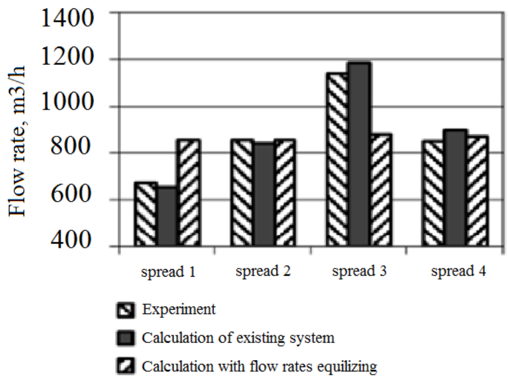


Fig. 3 The calculation results of the gas removal system.

Exhaust duct network and the geometry of the collecting drain were constructed according to the design drawings.

Network elements of the hybrid model are divided into three groups:

1. Edges, whose parameters are defined based on the geometric characteristics (edges 2, 9, and 10).
2. Edges, which model the area for installation of the regulating gate valve (edges 1, 4, 6, and 8). Minimal length and resistance correspond to the opened gate valve.
3. Edges describing the spread of below building exhaust duct (edges 0, 3, 5, and 7).

The parameters of the third group edges were determined based on the experimental data.

The results of the hybrid calculation were compared with experimental data. The rarefaction at the outlet of the collecting drain amounted to 2416 Pa. After the equalization procedure of the flow rates from electrolysis baths, that is, identification of the additional resistance, a re-calculation was carried out, whose results are presented in Fig. 3. The rarefaction in the system required for obtaining the desired gas flow rates from the baths, rose up to 2868 Pa.

2.3 A hydrodynamically disconnected model

When modeling objects, which are characterized by hydrodynamically disconnected or loosely connected flow of several fluids, we can use the third method to build multi-scale models. In this kind of models the network one-dimensional elements "penetrate" spatial area, for example, tube bundles in the heat exchanger. The connection between the network and spatial parts of the model is carried out through the source terms of the transport equation.

When calculating tube-and-shell heat exchangers by empirical formulas, a number of problems arise due to the neglect of the uneven distribution over the heat exchanger volume of not only temperature, but physical properties of the coolant. This may lead to incorrect determination of themal head and the hydraulic resistance of the tube bundle and consequently result in incorrect calculation of heat transfer and pressure drop.

When modeling tube-and-shell heat exchanger, it was necessary to carry out calculation of tube bundle consisting of six hundred tubes that would require the use of a computational grid consisting of about hundreds of millions of cells, and therefore the use of large computing power. When modeling the heat exchanger, employing the hybrid technique, each passage of the pipe from one tube board to the other is represented as a separate network element. A hybrid model of interaction between network and spatial elements allows calculating the heat fluxes and resistance coefficients, determining the temperature fields (Figs. 4, 5) and the substance properties, as well as the pressure drop in the system.

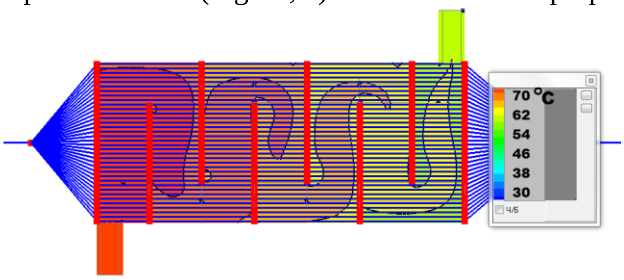
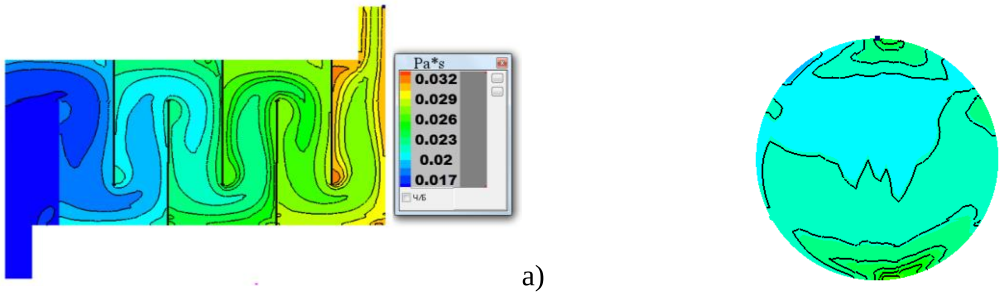


Fig. 4 Temperature field in the central cross-section of the multi-scale heat exchanger model



a) b)

Fig. 5 Viscosity field in the heat exchanger cross-section:
a) longitudinal cross-section; b) transverse cross-section.

At that, for the simulation we used the spatial grid consisting of 125,000 cells and a network model consisting of 25,000 edges, describing the pipelines that allowed performing calculations by means of a personal computer.

2.4 Combined model

The above described models can be combined. An example of a combination of hydrodynamically unified model and disconnected model is demonstrated on the example of heat transfer calculation in the cooling unit of the electronic chip with branched microchannel flow system. This is the conjugated heat transfer problem in the system with different spatial scales. The combination of the two methods consists in the fact that on the one hand the hydraulic part of the task consists of both spatial and network elements (channels), which requires the associated description, while on the other hand, thermal part of the problem describes the interaction between the cooling fluid and a chip through the energy exchange using hydrodynamically disconnected model. Comparison of the results was carried out based on the experimentally obtained temperature field and fully-spatial and hybrid calculations (Fig. 6). Quantitative verification of temperature distribution along the two given lines is shown in Fig. 7. The result of the comparison shows the adequacy of the application of the hybrid approach for solving this class of problems.

Conclusions

Despite the significant increase in computational power, many complicated problems of hydrodynamics and heat transfer cannot be resolved based on solving spatial task only at the expense of detail grid partitioning, even with wide-sweeping parallelization that is just extensive approach. One type of intense approach is the application of hybrid models. In this case, depending on the level of necessary information, different task areas are calculated using various models with different spatial resolution. The paper discusses ways of implementing hybrid models.

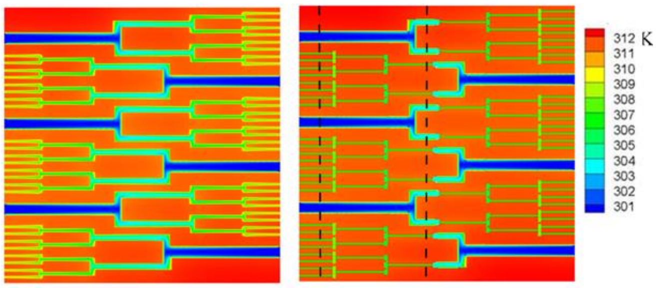


Fig. 6. Comparison of temperature fields in the central cross-section:
a) spatial calculation; b) hybrid calculation.

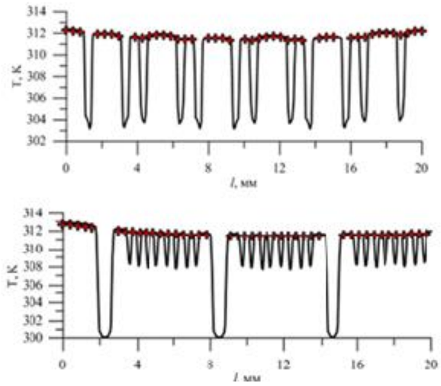


Fig. 7. Comparison of the temperature in two cross-sections highlighted by the dotted line in Fig. 6:
a) left cross-section, b) central cross-section; solid line shows spatial calculation, while markers correspond to hybrid calculation.

References

[1] Merenkov AP and Khasilev VYa. Theory of hydraulic circuits. Moscow, "Nauka", 1985.
[2] Atavin AA. Calculation of transient flow of water in ramified systems of riverbeds or channels. Collected papers: Dynamics of a continuous medium. 1975, No. 22. pp. 25-36.
[3] Wilson R. Introduction to graph theory. Moscow, Mir, 1977, 208 p.
[4] Patankar C. Numerical heat transfer and fluid flow. Moscow, Energoatomizdat, 1984.
[5] Fletcher CA. J. Computational techniques for fluid dynamics. Moscow, Mir, 1991, Vol. 1-2. P. 1054.
[6] Novitsky NN, Sukharev MG, Tevyashev AD, et al. Using an integrated approach in the study and optimization of exhaust duct operating modes. In the book: Energy pipeline system: Methodological and applied simulation problems. Novosibirsk, Nauka, 2015. p. 476.
[7] MpCCI 4.4.2-1 Documentation. 2015.
[8] Wedding WC. Multiscale modeling of the mine ventilation system and flow through the gob. Ph.D. thesis in Mining Engineering. University of Kentucky, 2014.
[9] Dobroserdova, T.K. Numerical simulation of blood flow in the presence of vascular implants or pathologies. Ph.D. thesis in physics and mathematics, 2013.
[10] Mikhienkova EI, Filimonov SA., and Dekterev AA. A hybrid algorithm for the joint calculation of multi-scale hydraulic problems taking into account thermal processes. Computational methods and programming: new computing technology, 2016, Vol. 17, No. 4, pp. 380-392.
[11] Shakhrai SG. Enhancing the efficiency of the ventilation in buildings at aluminum electrolysis production by improving the gas removal system. Ph.D. thesis in engineering, 2008.

The reported study was funded by Russian Foundation for Basic Research, Government of Krasnoyarsk Territory, Krasnoyarsk Region Science and Technology Support Fund to the research project №16-48-242085