Numerical modelling and analysing of conjugate radiation-convective heat transfer of fin-tube radiator of spacecraft

A Delcov, A Hodenkov and D Zhuikov
Reshetnev Siberian State Aerospace University, Krasnoyarsk, Russia

E-mail: delkov-mx01@mail.ru

Abstract. This paper covered the problem of assessing the effectiveness of the section of the fin-tube radiator of space thermal control system. The task of calculating the conjugate radiation-convective heat transfer is presented. The results of numerical simulation are described.

1. Introduction
Currently there is the problem of calculating the conjugate radiation and convective heat transfer characteristic for a wide range of technical problems (the nozzles of rocket engines, industrial furnaces, chemical reactors, etc.). For this heat exchange contact the current environment and the heat-conducting surfaces as well as the presence of significant temperature gradient between the surface and the environment are the defining conditions.

The mathematical description of this problem causes some difficulties: the need to localize the heat transfer equations for the two areas – liquid and solid – followed by a joint decision. In this paper we consider the heat radiation panel on the spacecraft designed to remove heat energy into space. In space systems such panels are an important part of the thermal power system (for example, solar power plant or thermal control system) [1,2].

Radiation panel is one of the important elements of the thermal control system. In terms of space radiation discharge is advantageous way to remove heat from the spacecraft [3]. Consider the section of the radiator (Figure 1), which is designed as a pipe with a coolant being connected to the radiation plate.

On the one side of the section contacts the coolant flow is correspondingly the heat transfer from the coolant in the wall; the other side is that a panel of radiation with which the heat flow is discharged into the open space. Sections are grouped in panel radiators.

The problem of calculating the conjugate heat transfer in this case is divided into two sub-tasks: the temperature distribution in the solid (radiator) and the temperature distribution in the liquid coolant. These two problems are solved together with the general boundary condition – convective heat transfer through the wall of the pipe.

2. Governing equations
This problem is described by the equations of heat and mass transfer [4,5]. The major difference of this problem is that we have two domains – solid domain and fluid domain.
In the solid domain we have heat equation:
\[ \rho c_v \frac{\partial T}{\partial t} = \nabla \cdot (k \nabla T) \]  

The boundary conditions for the present fin geometry (fig. 1) are defined as:
for convection surface: \( k \frac{\partial T}{\partial n_{\text{tube, surf}}} = q_c = \alpha (T_{fl} - T) \)
for radiation panel: \( k \frac{\partial T}{\partial n_{\text{surf,A}}} = q_c = \varepsilon \sigma (T^4 - T_{out}^4) + q_{solar} \)
for adiabatic walls: \( k \frac{\partial T}{\partial n} = 0 \)

In this equations \( \rho \) – density of solid domain, \( c_v \) – heat capacity, \( T \) – temperature, \( t \) – time, \( k \) – thermal conductivity, \( \alpha \) – heat transfer coefficient, \( n \) – normal to surface, \( \varepsilon \) – emissivity, \( \sigma \) – the Stefan–Boltzmann constant, \( T_{fl} \) – temperature of incident flow in convection heat transfer, \( T_{out} \) – temperature of surrounding area in the radiation heat transfer, \( q_{solar} \) - solar flux.

In the fluid domain we have three equations:
\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0 \] (continuity equation)  
\[ \rho \left( \frac{\partial u}{\partial t} + u \cdot \nabla u \right) = \rho g + \nabla p \] (cauchy momentum equation)  
\[ \rho \frac{dh}{dt} = \nabla \cdot (k \nabla T) + \rho \frac{dq}{dt} \] (energy equation)

The boundary conditions for the present fin geometry (fig. 1) are defined as:
at x=0:  
\[ T = T_{fl} \]  
\[ v = v_{fl} \]  
\[ k \frac{\partial T}{\partial n_{\text{tube, surf}}} = q_c = \alpha (T - T_{solid}) \]

Empirical correlation for the Nusselt number is used to the determine heat transfer coefficient \( \alpha \):
\[ Nu = 0.33 \text{Re}^{0.5} \text{Pr}^{0.43} \] (laminar)
\[ Nu = 0.021 Re^{0.8} Pr^{0.43} \] (turbulent) (6)

where \( Nu = \alpha dk^{-1} \).

In this equations \( \rho \) – density of fluid domain, \( t \) – time, \( u \) – velocity, \( T \) – temperature, \( k \) – thermal conductivity, \( h \) – enthalpy, \( q \) – internal heat source, \( p \) – pressure, \( Re \) – Reynolds number, \( Pr \) – Prandtl number.

3. Methods of numerical modeling

The equations for calculation of processes in liquid and solid areas recorded for the three-dimensional case [6]. Depending on the purpose of the simulation is possible to simplify the equations for one-dimensional and two-dimensional cases. To study the problem of radiation-convective heat transfer in the framework of this work were used three methods:

- An algorithm developed by the authors (1-d on two domains);
- Calculation using a package Syrthes (3-D on solid and 1-d of fluid);
- Calculation using a combination of packet Syrthes and Code_Saturne (3-D on the two domains).

For a quick evaluation of the parameters of heat exchange in the first approximation, the authors have developed a 1-D model of the process based on the finite volume method. Area calculation model is the contact liquid and a solid in which the temperature change occurs only in length. The calculation is carried out in two phases: the first (predictor) detects the temperature of the liquid and solid region, taking into account the thermal equilibrium of only two contact-ties and volumes. At the second stage of the calculation work together to solve two problems for liquid and solid (each sub-task the assumption of fixed boundary conditions). The second stage uses a stationary iterative method for minimizing the residual temperature areas. This algorithm is suitable only for preliminary parameter estimation coupled heat (e.g., in the case of multi-parameter optimization), and is ineffective for the qualitative evaluation of complex cases of heat exchange with the presence of complex geometry and boundary conditions.

To calculate the nature of the temperature distribution in the solid area was used package Syrthes [7]. This package allows modeling of heat transfer in solids with different boundary conditions, including the conditions of radiative heat transfer. The interaction of solid and liquid regions is conducted through convective heat transfer boundary condition recalculation of the flow temperature with floor length.

For complex cases, the dual radiant and convective heat exchange with the 3-D representation of the liquid and solid areas requires the use of CAE-packages. For reasons of availability, the authors used open packages Syrthes (solid domain) and Code_Saturne (fluid domain) [7], united to work together with a script language python. The main solver is Code_Saturne. It uses temperature distribution regions obtained in the previous step for each subsequent iteration.

4. Test Case

As part of this work as a test problem the problem was solved with the following boundary conditions:

- Working medium - iso-octane;
- Material of the radiator – aluminum;
- A pipe with a diameter of 12 mm (inner) and 13 mm (external) and a length of 0.5 m;
- Radiation pad size 0.5 * 0.1 m, thickness 2mm pad;
- Emissivity for surface A is 1, for the surface B is 0 (multi-layer insulation - MLI);
- Solar flux is 0.

For the numerical solution using CAE-packet computational domain was divided into subregions of a solid and a liquid. The calculation results are presented in Fig. 2-3. The temperature of the coolant at the outlet shows the average over the cross section.
Figure 2. The nature of the temperature distribution on the radiating surface at a flow rate of 0.05 ms⁻¹.

As it is seen from the transverse profile of temperature distribution, temperature divergence in the central part and on the boundary is 1.5 K. The magnitude of discrepancies depends on the conductivity of the radiator. Obviously, the smaller the gap means better performance of the radiator. To obtain a uniform longitudinal profile a number of sections are combined with common collector. The longitudinal profile of the temperature has the nature of the spline, which is explained by the finite thermal conductivity of the material of the battery. The temperature decreases along the length of the coolant flow as a result of the backlog.

Meet the challenges of the three methods showed good consistency. The accuracy of the first method in many ways determined by the errors in the reduction of geometry and boundary conditions to the 1-d mean (especially the complex profile of the pipe, which is replaced by an equivalent rectangle). It should be noted that the third method causes considerable time for calculation and is not very suitable for solving multiparameter optimization. The second method was used as a basis for further research (Syrthes-only).

Figure 3. Radiating surface temperature profiles across and along fin-tube radiator.

5. Results and discussions

The impact of speed and coolant temperature on the performance of the section was estimated. The thermodynamic properties from the database REFPROP were used for calculating the heat transfer coefficient of isoctane. Results of numerical modeling are shown in tables 1,2.

To evaluate the effectiveness we using [6]:

4
The result of numerical studies has shown that increasing the coolant flow rate calculation and the efficiency of the radiator (see Fig. 4). This is due to two factors: the increase in the coefficient of heat transfer by increasing the flow rate and a decrease in the temperature difference in the coolant inlet and outlet. The absolute value of the efficiency of the section also increases with increasing temperature of the coolant. The efficiency for the temperature +30 °C is more than for the temperature +60 °C. It is connected with the fact that the heat flow at +30 °C is more than correspondingly smaller temperature difference on the surfaces that directly affects the efficiency.

Figure 4. Comparison of the effect of various factors on the performance of the radiator.
6. Conclusion

It should also be noted that the increase in the flow of coolant in the thermal control system increases the size and weight of the coolant pumps. In general, the results showed the possibility of the numerical solution of coupled heat transfer area for the radiator. For solving the optimization problem for several parameters (the area of the panel, flow geometry) it must be simplified the consideration of the problem (for example, using 1-D algorithm described above).

References

[1] Massardo A F, Tagliafico L A, Fossa M and Agazzani A 1997 J. Propul. Power. 13(4) 560–4
[2] Siamidis J 2006 Heat rejection concepts for lunar fission surface power applications 4th Int. Energy Conversion Engineering Conference and Exhibit (IECEC) (San Diego: AIAA)
[3] Delcov A V, Hodenkov A A and Zhuikov D A 2014 12th Int. Conf. on Actual Problems of Electronics Instrument Engineering (APEIE) (Novosibirsk: IEEE)
[4] Modest M 2003 Radiative heat transfer (New York: Academic Press)
[5] Minkowycz W J, Sparrow E M and Murthy J Y 2000 Handbook of numerical heat transfer (Hoboken: John Wiley and Sons)
[6] Kamrava M H and Bazdidi-Tehrani F 2011 Applied Mechanics and Materials 110-116 23–8
[7] Peniguel C, Rupp I, Rolfo S and Hermouet D 2014 Conjugate heat transfer study of a wire spacer SFR fuel assembly thanks to the thermal code SYRTHES and the CFD code Code_Saturne Joint International Conference on Supercomputing in Nuclear Applications + Monte Carlo ed D Caruge et al (Paris: EDP Sciences)