UDC 536.2

NUMERICAL SIMULATION OF FLOW AROUND A TRANSVERSE TRANSFER COMPACT BEAM TUBE IN TUBE HEAT EXCHANGER

V. Gorobets, V. Trokhaniak

Bundles of smooth cylindrical tubes with staggered and location are widely used in a variety of heat exchangers and devices power plants. Review of the literature and the results of a significant number of experimental studies of heat hydrodynamic characteristics bundles of smooth tubes in crossflow, including research on heat transfer of staggered bundles at Reynolds numbers shows that such beams have a high heat output compared to the line bundles. However, they have a higher flow resistance compared with the line bundles.

It should be noted that the surfaces of the type used in shell and tube heat exchangers, leading to an increase in their weight and dimensions. One way to improve these characteristics is the use of fins and heat transfer intensifiers for convective surfaces. However, the use of finned surfaces and significantly increases the hydraulic intensifiers resistance paths of the heat exchanger and requires the use of pumping coolant pumps and higher power fans. A promising direction to reduce flow resistance and heat transfer enhancement in convective heat exchange surfaces is to use a smooth tube bundles with a compact configuration.

Therefore, the development of new designs of shell and tube heat exchangers, tube bundles with compact is relevant and needs to be addressed.

Consider tube heat exchanger with rectangular casing, which is placed bundles of pipes compact configuration of the cross flow. In the heat exchange device used is the location of pipes, in which neighboring tubes touch each other and are shifted relative to each other along the vertical axis at a distance K, where $0 < K < \sqrt{3}D/2$, and the distance C is subject to the condition $C \geq D + 5мм$, due to the fact that manufacturing technology is much more complicated beam at distances between the pipes less than 5 mm.

The simulation of heat and mass transfer in channels with a compact tube heat exchanger tube bundles placement using the software package ANSYS Fluent. In a mathematical model underlying Navier-Stokes equations and the equation of convective energy transfer. In the considered case the selected standard ke turbulence model (KES).

An velocity field, temperature and pressure in the channel of the heat exchanger and conditions of hydrodynamic flow in the channels and heat transfer processes in these channels. Proposed and developed a new design and tube heat exchangers with a compact arrangement of tubes in tube bundles.

Analysis of the velocity field shows that the maximum value of the flow rate observed at the side walls of the heat exchanger, and their value is twice the velocity in the shell side channels. In some locations channel air speed is 85 m / s, and the average air velocity in the narrowest cross section kanalppu is about 45 m / s. In areas of the channel that separates the three sections of pipe bunch stagnant zones occur at the last trumpet each beam. In addition, these zones are observed in areas curved channel for individual elements tube bundle.

As seen in the top of the pipe is lead the boundary layer, and at the junction of adjacent tubes observed stagnation zone. In these areas there are two vouchers vortices whose flow rate is significantly lower than in the main stream.

It is interesting to compare the dimensions and weight of a new design of heat exchangers with known structures. Comparable dimensions and weight of the proposed design of shell and tube heat exchangers with a staggered arrangement of the corridor or pipes and heat exchangers, which form a continuous series of pipes that come into contact with each other and are not offset adjacent tubes. A comparison of weight and size of the new heat exchanger design and the above structures have the size and weight reduction of up to 60% for the same heat