Influence of condenser design on flow pattern and performance of shell-and-tube condenser for zeotropic refrigerant mixture

Dave Sajjan and Lennart Vamling

Department of Heat and Power Technology,
Chalmers University of Technology, SE-412 96 Göteborg, Sweden
Dave.sajjan@hpt.chalmers.se and lennart.vamling@hpt.chalmers.se

Abstract—A two-dimensional model has been developed to simulate the condensation of multi-component mixtures or pure fluids on the shell-side of a shell-and-tube type condenser. The flow pattern of the shell side gas has been calculated using a computational fluid dynamics (CFD) program.

A standard pure fluid condenser design is used as reference and the effects of geometrical modifications, i.e. reduced shell-side by-pass, making the impingement plate porous and using baffled shell, on the condenser performance are investigated for the zeotropic mixture R407C. The effects can be summarized as follows:

- By using a porous impingement plate and decreasing the shell-side by-pass, the condenser performance increases by around 12% compared to the base-case configuration.
- The geometrical modifications have higher impact on condenser performance at low temperature driving force, i.e. low temperature difference between the bubble point and the coolant temperature.
- The baffled shell where the condensate is separated from the gas in each baffle section can lead to very poor condenser performance due to increasing concentration of lighter components in gas phase.

Introduction

The unexpected discovery of the effects of chlorofluorocarbons (CFCs) and hydro-fluorocarbons (HCFCs) on the ozone layer has pushed the refrigeration industry to find alternative, environmentally friendly refrigerants. At present, R22 (HCFC with chemical formula $CHClF_2$) dominates as the most widely used refrigerant in Heating, Ventilation, Air Conditioning and Refrigeration (HVAC&R) industry. Since no single fluid meets all the economic, environmental and non-toxic criteria as a replacement for R22, blends of different substances are under consideration. One such mixture is R407C

(38% by mole R32, 18% R125 and 44% R132a), which can be used as a replacement for R22 also in the existing machines after some changes.

Experimental results after such a replacement show that condenser performance may fall severely at low condensation loads compared to R22 [1]. Possible reasons for this performance deterioration were studied in [2]. It was seen that the kind of condensation curve, i.e. differential or integral, was a factor influential enough to explain the differences in the performance of R407C relative to R22. The differential kind of condensation occurs when the concentration in the condensate at the gas/condensate interface is not influenced by the draining condensate from the tubes situated above. The differential kind of condensation may occur due to one or the other of the following reasons:

1. The condensate removal from the system takes place immediately after it is formed.
2. Infinite mass transfer resistance in the condensate film.

The other extreme, integral condensation, occurs when there is perfect mixing between the new and the old condensate and the above-described criteria are not fulfilled. The effects of the kind of condensation curve on condenser performance are studied in [2].

It is concluded in [2] that, to predict the performance precisely and to understand the problems involved in the condensation of mixtures with glide in shell-and-tube type condensers, it is important to examine the detailed processes occurring around each of the tubes. Gas flow pattern is one of the important parameters since the local concentration of the gas, and consequently of the condensate at any particular point, is dependent on the path along which the vapour reaches that point. Investigations regarding the performance of condensers for mixtures [4, 5] have shown that the amount of vapours vented with non-condensable gas, and the position of the vent, are important factors. This means that building pockets of accumulated non-condensable gas or lighter components in the case of zeotropic mixtures would most probably occur in areas where gas velocities are low and a proper mixing in the gas bulk cannot take place. Such pockets with accumulated will decrease the performance of the tubes in these areas.

Traditionally, condenser designs are based upon empirical information and assumed flow fields. The designer arranges condenser tubes to give a satisfactory performance. A shell-side by-pass is used to facilitate access of vapour to the lower region of the tube bank without excessive pressure drop. In general we can say that the design is optimized to achieve maximum heat transfer at low-pressure drops. But in the case of zeotropic mixtures the condensation temperature, pressure and the concentrations of the components vary spatially. If the design is optimized only for low-pressure drop as for pure fluids, it may not be an optimized design for zeotropic mixtures. Key issues for the design of condensers for zeotropic mixtures can be summarized thus:

1. achieve low pressure drop,
2. avoid any dead zones,
3. avoid segregation of the gas and the condensate.

Points 1 and 2 are important for both the pure fluids and the mixtures, but effects of gas and condensate segregation, point 3, in the case of zeotropic mixtures are important due to the reasons mentioned above.

The possibilities of using CFD in condenser design

The condensation processes are a complicated combination of two-phase flow, heat and mass transfer. Introducing blends with temperature glide complicates the scenario. Methods for experimental determination of concentration variation in a condenser have been discussed in literature [3]. There, the distribution of the concentration is determined from the measured static pressure and local temperature, using phase equilibrium diagrams. This kind of methodology is, however, directly applicable only to binary mixtures.

Computational Fluid Dynamics (CFD) provides a very useful tool to understand flow patterns and their influences on phenomena like heat, mass transfer and chemical reactions etc. With CFD it is possible to include e.g. the complex geometry, flow pattern, and physical processes like heat and mass transfer and pressure drop in a single model. Once the problem is modelled, CFD can be very effective to perform design optimization studies.

Aim

The aim of the paper is to investigate how appropriate a standard pure fluid design is for a zeotropic mixture and how the performance can be improved by different geometrical modifications. Possible modifications are:

1. reduced shell-side by-pass,
2. making the impingement plate porous,
3. introduction of baffles,
4. reduction of tube pitch.

In the present paper we try to estimate the effects of the modifications 1, 2 and 3 by using CFD and a computer program developed at our department. The motivation for reducing the shell-side by-pass and making the impingement plate porous is that it might lead to improved and more uniform flow pattern. By using baffles, higher gas velocities can be achieved which would lead to lower mass transfer resistance and higher vapour-side heat transfer.

General features of CFD

CFD modelling is a computational technique to simulate fluid flow, heat and mass transfer by solving Navier-Stokes, thermal-energy and species equations iteratively. These equations are usually written as partial differential equations

$$(\vec{U}, P, T, \Phi) = f(x, y, z, t)$$

where $\vec{U}$ stands for velocity vector (u, v, w), P for pressure, T for temperature and Φ for some scalar that has to be calculated, as functions of independent space variables (x, y, z) and time t. Because the resulting equations are highly non-linear, they are not solvable by explicit methods. Approximate methods such as finite-difference, finite-volume or finite-element methods are used. In all these approaches, the physical domain, shown in Fig.1, is divided into computational cells as shown in Fig. 2. The number of cells depends upon the size of the physical domain and the required flow details. By modelling the small-scale mechanisms of the condensation processes within the framework of large-scale flow calculations, the problem can be handled very effectively. Flows for more than one phase can be simulated in CFD packages, but the complexity of the models and the computational time required may be too high.

Simplifying assumptions and the calculation procedure

Modelling shell-and-tube condensers by taking two-phase flow, heat and mass transfer as well as complete geometry into account requires excessive computing capacity. To get reasonably short computational time, we have chosen to make some assumptions.
The present model is two-dimensional, i.e. only a slice of the condenser is considered. In order to utilize symmetry, the condenser inlet is assumed to be located at the top middle of the condenser. This leads to the computational domain shown in Fig. 1.

Modelling processes like two-phase heat and mass transfer requires very high grid density. To avoid such high grid densities the problem is solved as one-phase with condensing component fluxes as boundary condition. Only the gas bulk is simulated with the CFD program [7] and the condensation processes around each tube are modelled with User Defined Subroutines linked to the main CFD program. Tube walls in the CFD program are treated as gas-condensate interface. Gas-bulk temperature, species concentration and velocity from the CFD program are passed to the User Defined Subroutine, and the calculated condensing mass fluxes and the condensation temperature assuming an integral kind of condensation in the Subroutine are specified as boundary conditions in the CFD model. The whole procedure is an iterative loop where boundary conditions are modified on the basis of the bulk conditions in each sub-loop. The equations solved are given in [2].

Calculations for the baffled shell are performed by using the simulation program developed at the department [2]. Here all tubes in a row are assumed equal, making these calculations also two-dimensional.

Results

The main objective of this paper was to investigate the effects of flow pattern on the shell-and-tube condenser performance for a zeotropic mixture, R407C. Different geometrical modifications that can affect the flow pattern are studied, and the condenser performance is compared to the performance for the base-case geometry.

Effect of introducing baffles

Baffles can be used to increase the velocity of the shell-side fluid as well as to mechanically support the tubes against flow induced vibrations. The most common baffle type is segmental baffle, with a baffle cut resulting in a baffle window. The lower edges of the baffles are usually notched to permit draining.

The possibility of using a baffled shell for total condensation of R407C was investigated by using the simulation program presented in [2]. Totally, three baffle sections were used with decreasing spacing; the condensate was assumed to separate at the bottom of each baffle section, and the gas flow to reverse from one baffle space to the next. Figure 3 illustrates the change in the condensation temperature due to gas-condensate segregation at the baffle end. In the course of calculation we have seen that the use of baffles increases condensation rates significantly in the first baffle space, but segregation of gas and condensate leads to such lower temperature that the significant part of the last two baffle spaces is used to condense the gas with high R32 concentrations. A conclusion that can be drawn on the basis of this calculation is that the use of baffles, which can lead to gas-condensate segregation is something that should be discouraged for the total condensation of multi-component mixtures with glide.

Effect of modified shell and porous impingement plate

The effects of geometrical modifications on condensation rates compared to the base-case configuration are presented here. The following geometrical modifications are investigated:

1. base-case configuration with porous impingement plate,
2. modified shell, i.e. decreased shell-side by-pass,
3. modified shell and porous impingement plate.

Effect on flow pattern

An impingement plate opposite to the gas inlet is used to reduce tube erosion or vibrations. It may, however, deflect the flow so that a significant amount of gas flows through the shell-side bypass. The result can be that a significant part of the condenser remains ineffective because gas bulk has to take a long path to reach the tubes in the middle of the tube bank. In this work, the effects of using a porous impingement plate such that about 50% of the inlet gas flows through it are investigated. Figures 5 and 6 show the streamlines, which are lines that are everywhere tangential to the local velocity, of the gas velocity for the base-case condenser configuration and for the one where the plate is changed to a porous plate, respectively.

Shell-side by-pass is provided to facilitate access of vapour to the lower regions of the tube bank without excessive pressure drops. If the by-pass is too wide it can lead to lower gas velocities, and consequently to lower condensation rates on the tubes deep in the tube bank.

The heavier component, R134a in the present case, is more liable to condense first. This can lead to higher concentration of R32 in the areas where gas velocity is low. Hence, a more uniform velocity profile would counteract the accumulation of R32. The possibilities of decreasing the shell-side by-pass and also using porous impingement plate were investigated. The flow pattern for the modified shell with porous impingement plate is shown in Fig. 7. As we see, a smaller shell-side by-pass and the use of porous plate give a uniform velocity profile.

Effect on overall performance

Figure 4 compares the effect of the geometrical modifications on the overall performance of the condenser. There, the ratio of the total condensed mass flux to the condensed mass flux for the base-case configuration is shown.

As can be seen in Fig. 4, the condensation rate for R22 is almost unaffected for all modifications, but the condensation rate for R407C increases significantly when the gas flow is more uniform, i.e. in the condenser with small shell-side by-pass and porous plate. We see that the condensation rate increases by around 12% at low temperature driving force, i.e. the temperature difference between the bubble point and the coolant temperature and by around 8% at high temperature driving force.

Effect on species concentration

Species concentration depends upon both the diffusional and the convective mass fluxes. In the areas where the gas bulk has difficulties in penetrating, accumulation of R32 can take place. The variation of the R32 concentration for the base-case condenser configuration along a horizontal plane (tube row) is shown in Fig 8. The tube row numbers, shown in Figs. 8 and 9, increase from the inlet to the outlet; the first tube row is opposite to the inlet, and the horizontal distance from the symmetry line to the shell is plotted along the x-axis.

The effect of modifications on the mass fraction of R32 in the gas bulk is shown in Fig. 9. As can be seen, the concentration of R32 becomes more uniform for the modified shell with porous plate.

Uncertainties/assumptions

A grid independence check was performed by doubling the grid density (to 250×300 from 125×150) and it was found that the condensation rate increases for the finer grid by 5%. We assume here that the simulations are reasonably grid-independent.

Gas flow is equally spread out along the entire tube length, reducing the calculations to two dimensions.

Equal condensation rate around the tube surface was assumed, even though we are aware that the local calculation around the tube surface, for both heat and mass transfer, would probably give better estimates of the condensing fluxes. These simplifications are needed, first, because most of the correlations used to calculate the heat and mass transport are based on average gas flow and give average values, and second, to get reasonably short computational time.

The heat transfer coefficient for the condensate layer is calculated using the Beatty and Katz [6] equation developed for low-finned tubes. This model does not account for the drainage due to surface tension, but the surface tension effects on condensate drainage for 3-D pin-finned tubes could be more profound**.**

The obstruction to the gas flow due to condensate film is not taken into account; the only obstruction to the gas flow that is taken into account is due to tubes.

Conclusions

A two-dimensional model has been developed to simulate the condensation of multi-component mixtures or pure fluids on the shell-side of a shell-and-tube type condenser. The flow pattern of the-shell side gas has been calculated using a computational fluid dynamics (CFD) program.

The effects of geometrical modifications on the condenser performance are investigated. From these investigations the following conclusions can be drawn:

- The condenser performance increases by around 12% for the condenser with porous impingement plate and decreased shell-side by-pass compared to the base-case configuration.
- The geometrical modifications have higher impact on condenser performance at low temperature driving force, i.e. difference between the bubble point and the coolant temperature.
- A baffled shell where the condensate is separated from the gas in each baffle section can lead to very poor condenser performance due to increasing concentration of lighter components in the gas phase.

References:

1. Gabrielii, C., Vamling, L., "Replacement of R22 in tube-and-shell condensers: Experiments and simulations", International Journal of Refrigeration, 1997, Vol. 20, pp. 165-178.
2. Sajjan, D., Vamling, L., Reasons for drop in shell-and-tube condenser performance when replacing R22 with R407C".
3. Fujii Tetsu, "Overlooked factors and unsolved problems in experimental research on condensation heat transfer", Experimental Thermal and Fluid Science, 1992, Vol. 5, pp. 652-663.
4. Hampson, H., Thain, W., "The effect of venting arrangements during condensation on a horizontal tube from a static vapor gas mixture", I.MECH.E., Proc. 3rd International Heat Transfer Conference, Chicago, 1966.
5. Furuya, Tai, Noji, Nobuharu, "Applications of non-azeotropic refrigerant mixture in prototype heat pump", Condensation and Condenser Design, ASME 1993, pp. 273-280.
6. Beatty, K.O., and Katz, D.L., "Condensation of vapors on outside of finned tubes", Chem. Eng. Prog., 1948, Vol. 44, pp. 55-70.
7. Fluent Inc., version 4.2.

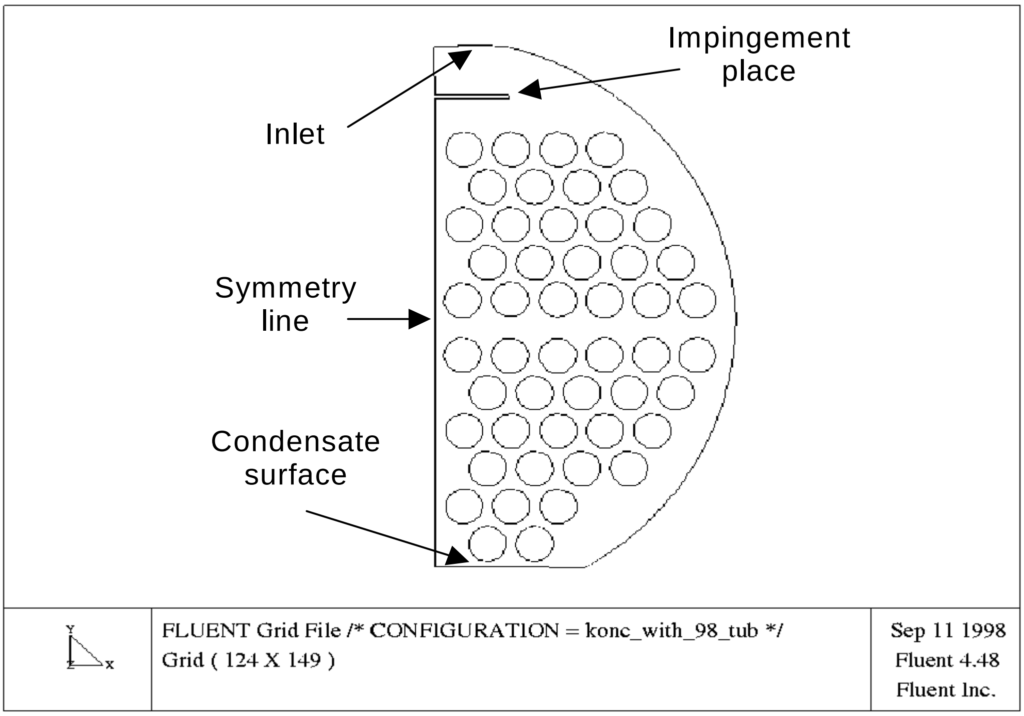

Figure 1: Condenser geometry.

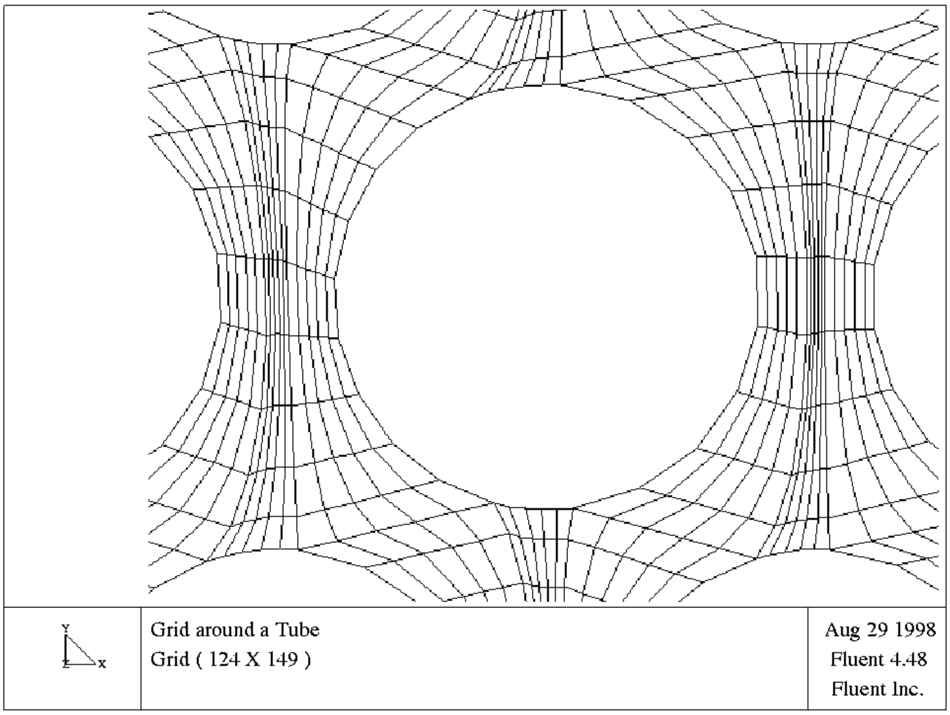

Figure 2: Computational grid around a tube.

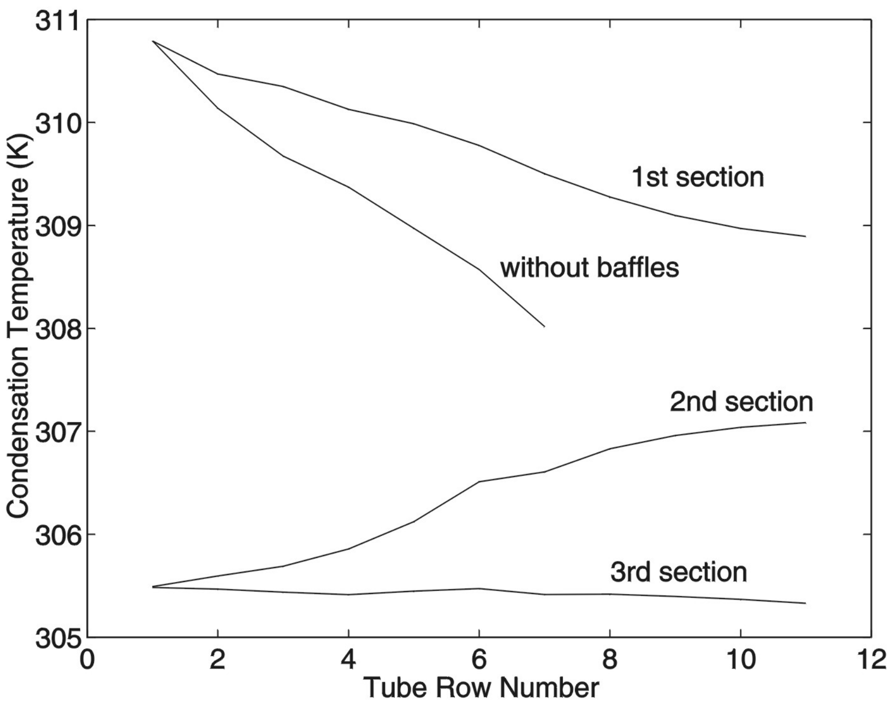

Figure 3: Variation of the condensation temperature for single shell and for the same shell with three baffle sections.

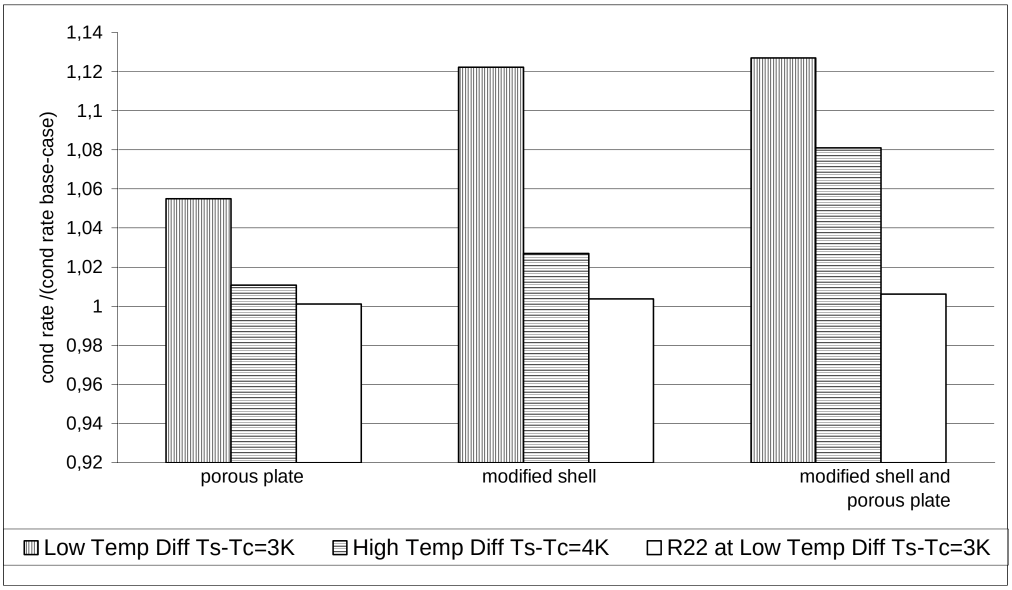

Figure 4: Change in the condenser performance for different geometrical modifications.

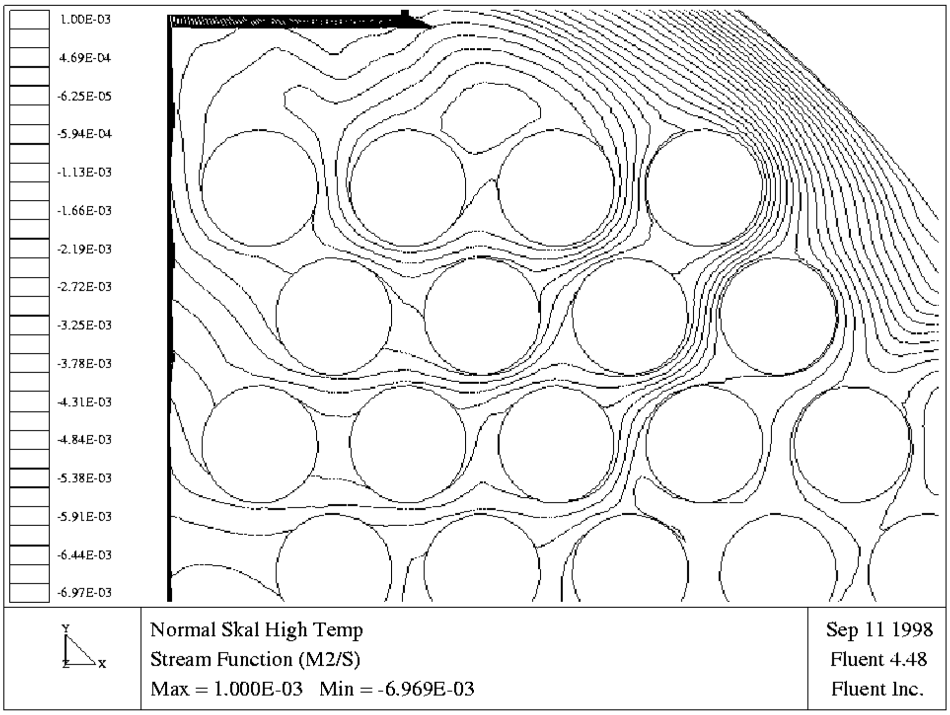

Figure 5: Streamlines for the ”base-case geometry” at higher coolant to gas temperature difference.

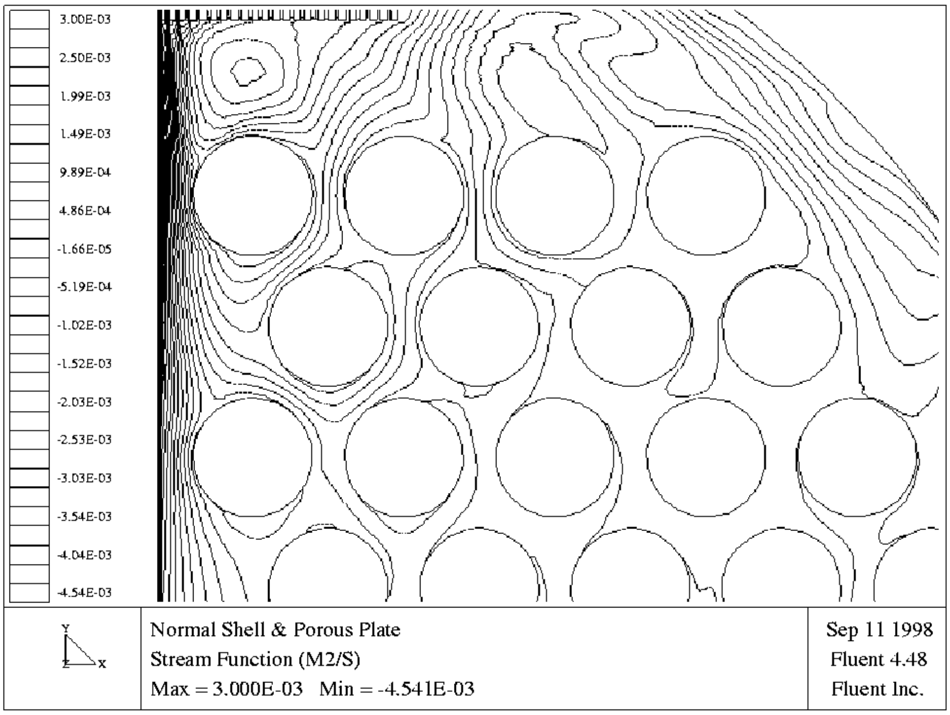

Figure 6: Streamlines in case of the porous plate.

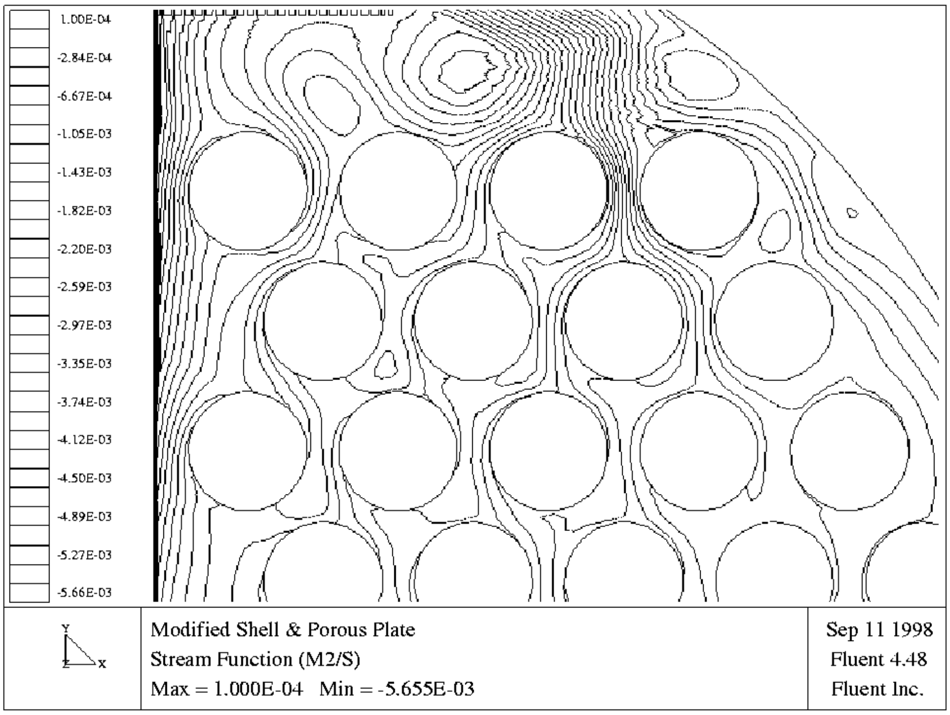

Figure 7: Streamlines for the case with decreased shell-side by-pass porous plate.

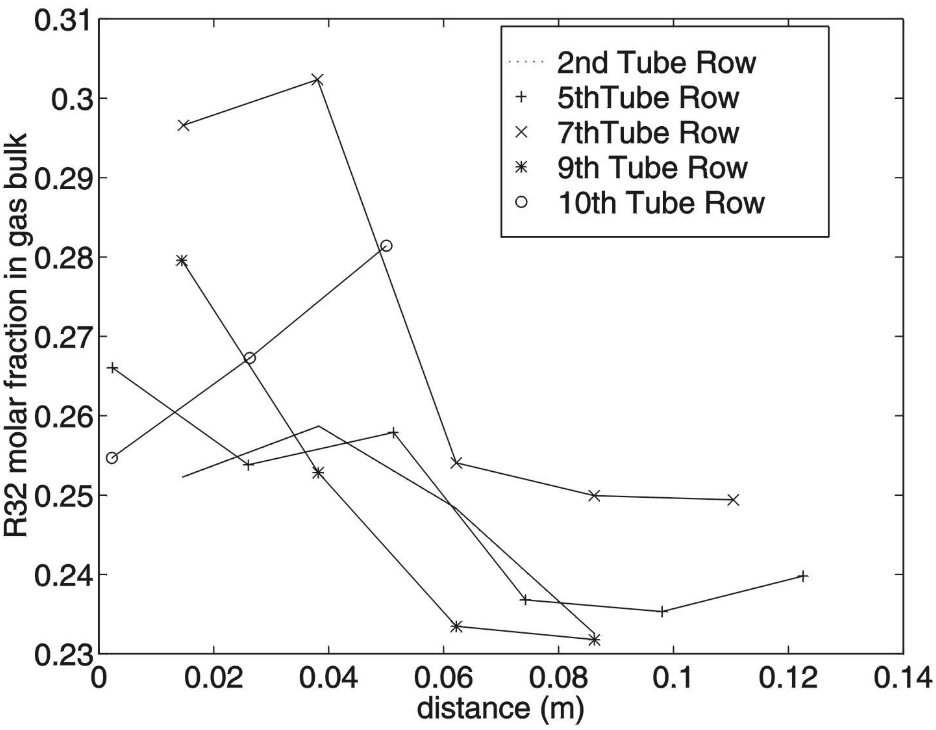

Figure 8: Variation of R32 molar fraction along tube length for "base-case geometry" (starting from the symmetry line in Fig. 1).

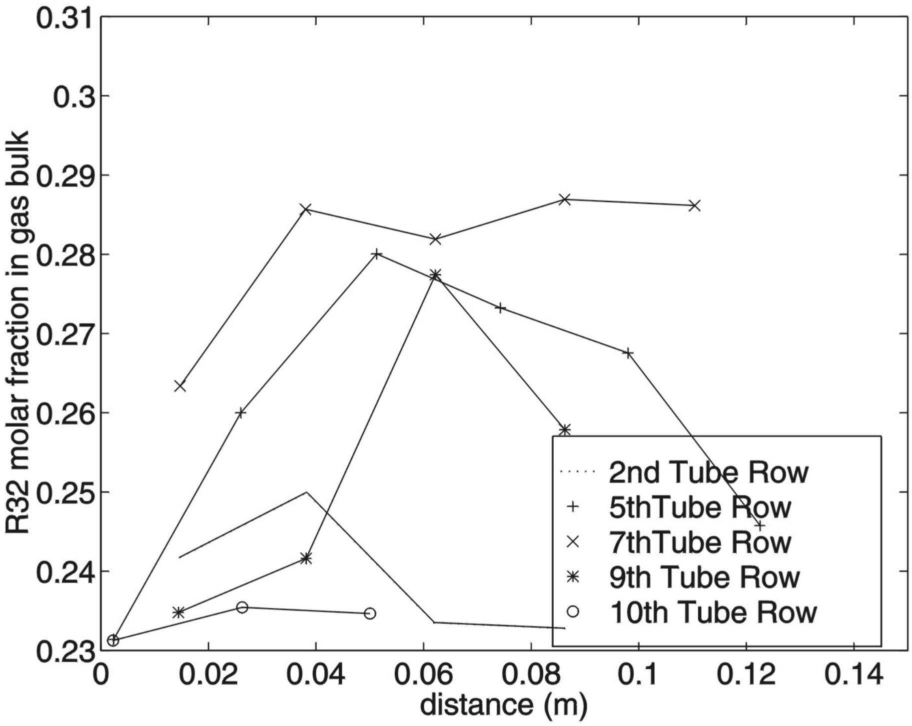

Figure 9: Variation of R32 molar fraction along tube-length for the case with decreased shell-side by-pass.