NENE 2022
September 12-15
PORTOROŽ
SLOVENIA
31st International Conference Nuclear Energy for New Europe

Design Optimization of Heat Transfer Performance in the Heads of Flow Boiling Experiment

Anil Kumar Basavaraj
Reactor Engineering Division, Jožef Stefan Institute
Jamova cesta 39
1000, Ljubljana, Slovenia
anil.basavaraj@ijs.si

Boštjan Zajec, Blaž Mikuž
Reactor Engineering Division, Jožef Stefan Institute
Jamova cesta 39
1000, Ljubljana, Slovenia
bostjan.zajec@ijs.si, blaz.mikuz@ijs.si,

ABSTRACT

Flow boiling is an effective heat transfer mechanism, which is important in many industrial applications including in nuclear power plants. A unique flow boiling experiment has been constructed in Thermal-Hydraulics Experimental Laboratory for Multiphase Applications (THELMA) at Reactor Engineering Division of Jožef Stefan Institute. The experiment consists of a custom-designed heat exchanger, which allows visual observation of the boiling surface. Heat flux at the boiling surface is controlled with the temperature and flow rate of the two fluids involved. The present design provided accurate measurements for low and medium heat flux magnitudes, however, modifications are needed for the flow boiling studies at high heat fluxes.

Numerical simulations provide better understanding of the complex devices as well as enable their design optimization. The main objective of the present study is optimization of the heat exchanger, which is used for flow boiling experiments. In particular, previous studies have shown that up to 50 % of the total heat transfer in our experiment takes place in the inlet and outlet manifolds, i.e. the heads of the heat exchanger. In order to increase the heat transfer in the test section itself, all heat losses need to be reduced to the minimum, including the heat transfer in the heads of the heat exchanger. For that reason, a computational fluid dynamics (CFD) model has been constructed for the present heat exchanger, which includes conjugate heat transfer in the primary and secondary fluid flow as well as several solid domains that are made of different solid materials. Results revealed the most critical parts of the device with severe heat leakages, which need improvements. Thus, modifications have been proposed in the geometry as well as in the selection of more appropriate material properties. Comparison between the present and the optimized design has shown significantly better heat isolation of the fluids inside the heads, which will hopefully allow experiments at much higher heat fluxes up to the critical heat flux (CHF).

Keywords: Heat transfer, CFD, Heat exchanger, Heat flux

INTRODUCTION

The application of heat exchangers is growing in great extent and pushing the urge to improve techniques for enhanced heat transfer. There have been many studies on the turbulent flow in a double pipe heat exchanger (DPHE), which improve heat transfer by three different methods: active, passive and compound methods [1]. In the active method, the heat transfer rate is increased by using external force, whereas in passive method surface or geometry modification stimulates the rate of heat transfer. The compound method is a combination of both the active and passive method. In the present study, we applied passive method with unique turbulators that enhance heat transfer. Namely, a fin type structure inside the inner pipe of DPHE helps enhancing mixing of the secondary flow, increasing heat transfer area and enhancing convective heat transfer by increasing turbulence.

In THELMA laboratory of Reactor Engineering Division, Jožef Stefan Institute, an apparatus has been built for visual observation of local heat transfer in wide range of single- and two-phase flows up to the critical heat transfer (CHF) as well as beyond it [2]. By designing the test section which resembles a close vicinity of a single fuel rod of the light water reactor, a better understanding of the CHF on a curved surface of the rod is possible [3]. In order to better understand the heat transfer inside the apparatus, a Computational Fluid Dynamics (CFD) analysis of the conjugate heat transfer in the secondary fluid was carried out to obtain local heat flux at the surface of the rod [4]. That model has been extended to include the primary fluid flow as well as the heat losses towards the ambient air [5]. Since the model consisted of only a few fins in the secondary fluid channel, periodic boundary condition was used for the refrigerant (primary) flow. The thermal boundary layer of less than 0.5 mm thick was observed at the solid-refrigerant interface and the axial pressure drop for the secondary flow has been estimated at the given flow rate [5]. In the next step, the whole measuring section with the inlet and outlet manifolds (the heads) were modelled and validated with the experimental measurements [6]. That CFD model allowed us to predict heat loss on the heads and evaluate the local surface temperature at the refrigerant walls along the entire measuring section.

Our present apparatus turned out to have large heat losses in both the heads up- and down-stream the test section. Since the heat transfer can be visually observed only in the transparent test section of the apparatus, it is highly desirable to reduce the heat transfer between both the fluid flows and particularly in the inlet head. Thus, in this paper, we analysed the present and a newly optimized design of the inlet head, which are shown in Figure 1 at the left and right side, respectively. Heat is transferred by convection with the fluid flow and by conduction in the solid regions. The purpose of the heads is to distribute fluids evenly. In co-current flow situation, the cold refrigerant and hot water flow come in contact with solid parts made of metal materials, which are good thermal conductors. Thus, large temperature difference drives high heat flux, which causes boiling of the refrigerant in the inlet head before entering the test section. For that reason, design optimization of both the heads have been proposed, which modified the geometry and selection of some materials. In particular, three main modifications have been proposed:

- The inner copper fins (orange color in Figure 1), which enhance heat transfer, have been removed inside the heads.
- The length and thickness of the inner tube (black color in Figure 1) which acts as structural support for the head part has been increased and the material has been changed from brass to plastic (Polyaryl Ether Ketone), which is thermal insulator.

- The refrigerant collector material has been changed from brass to stainless steel (yellow color in the right picture of Figure 1) to reduce heat conduction.

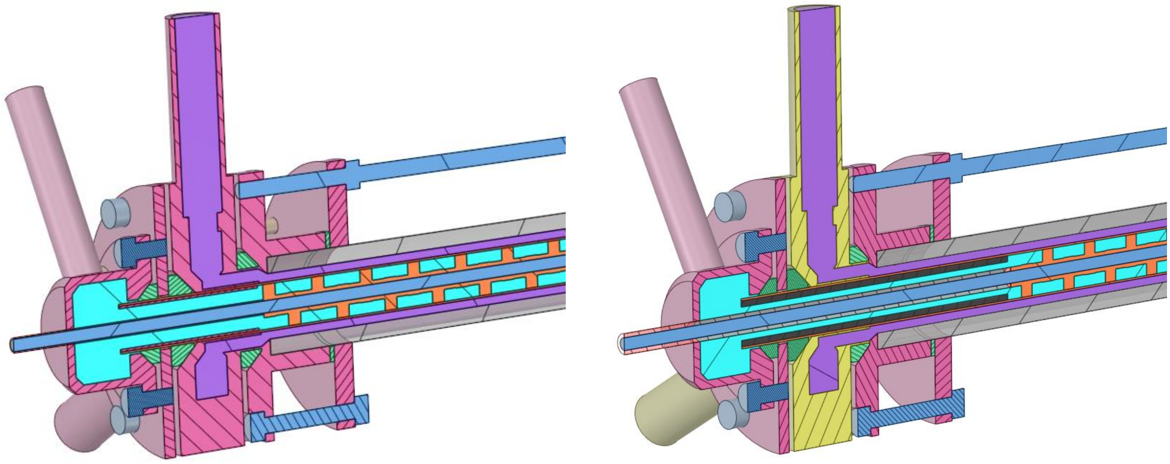


Figure 1: Cross section of the present (left) and optimized (right) geometry of the inlet head. Bright blue and violet colour correspond to water and refrigerant flow domain, respectively. Red, orange, yellow, grey, dark blue and black colour correspond to brass, copper, glass, stainless steel and plastic materials, respectively.

In this paper, we numerically evaluate the turbulent flow and heat transfer in the present and optimized design of the applied experimental device. The main objectives are the following:

- To study the effect of design optimization of the present geometry and selection of appropriate materials in a concentric double pipe heat exchanger,
- To reduce the heat loss in the heads and increase the heat transfer in the test section.

1 NUMERICAL MODEL

1.1 Geometry

The simulation was conducted using the 3D numerical model shown in Figure 1 and Figure 2. The test section consists of a forced circulation loop of cold refrigerant (R245fa) as primary fluid and hot liquid water as the secondary fluid. The test section was constructed to represent a double-pipe heat exchanger with finned structure in the inner tube to generate better mixing in the secondary fluid. The two identical heads on each side of the test section act as a connecting manifold. The primary fluid has three inlets and three outlets which are 120° apart for equally distributing the refrigerant around the axis before entering the test area. Contrarily, one single inlet and outlet on both sides are used for the secondary fluid flow, which runs through a harrowing path within the dedicated test section – a tube resembling a single fuel rod. A detailed explanation about geometry has been discussed in the previous papers [5] [6].

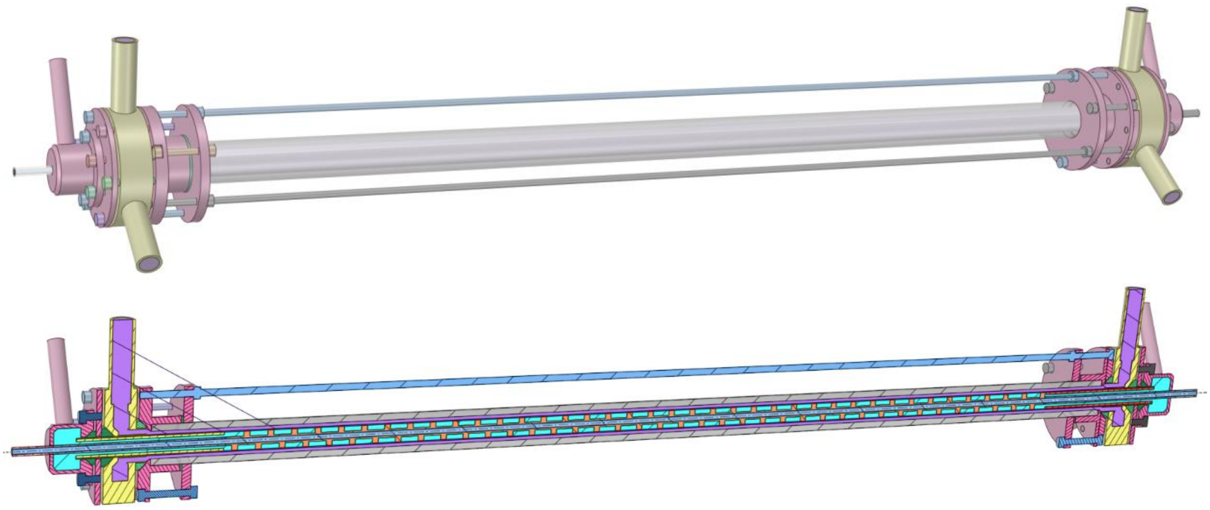


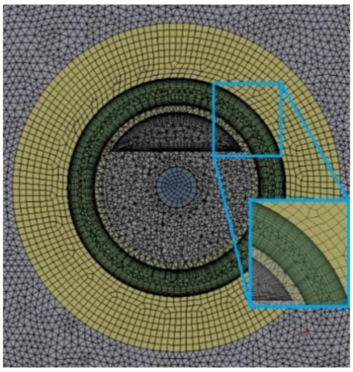
Figure 2: The optimized design model for analysis and it’s cross-section

1.2 Computational mesh

Mostly tetrahedral mesh was generated in the computational domain of the channel with the “ANSYS Meshing” tool. Near the walls, a prism layer meshing has been intensified in order to solve the boundary layer. Figure 3 shows the meshing techniques applied with the inflation layer. The unstructured mesh with fine meshing near the solid-fluid interfaces has been used, which resulted in a total number of around 24 million cells. The fully conformal mesh has been generated for the multi-body part (i.e. for the solids and fluids), which allowed that all the body interfaces were controlled automatically with the “Share topology” option. Furthermore, the chosen first layer at the walls ensures the y+ values are less than 1 and are given in Table 1. Mesh sensitive test has been performed and discussed in the previous paper [2].

Table 1: Parameters of mesh generation

Maximum tetra size [mm]		0.52
y^+	Liquid water [mm]	0.02
	Refrigerant [mm]	0.016
Number of prism layers	Liquid water	10
	Refrigerant	10
Growth rate	Liquid water	1.25
	Refrigerant	1.30



.
Figure 3: Mesh of the annular test section

1.3 Boundary conditions

Mass flow rates and inlet temperatures of both fluids are given in Table 2. Heat transfer coefficient of 5 W/m^2 K has been specified on outer surface of the model, which was in contact with the ambient air. The outer surfaces consist of metal materials in both heads and glass tube in the test section. The boundary conditions used in simulation are shown in Table 3.

Table 2: Measured data for non-isothermal case

	Water		**Refrigerant**		**Air**
Type	Tw [°C] inlet	Mass Flow Rate [kg/h]	Tr [°C] inlet	Mass Flow Rate [kg/h]	Ambient T_{air} [°C]
Non-isothermal	41.79	11.96	17.83	80.09	24.4

Table 3: Boundary conditions used in simulation. Prescribed temperatures and mass flow rates for each case are listed in Table 2.

<table>
<tr><th colspan="2"></th><th>Momentum</th><th>Temperature</th></tr>
<tr><td rowspan="2">Inlet</td><td>Liquid water</td><td>Prescribed mass flow rate</td><td>Prescribed constant T_{in}^{W}</td></tr>
<tr><td>Refrigerant</td><td>Prescribed mass flow rate</td><td>Prescribed constant T_{in}^{R}</td></tr>
<tr><td rowspan="2">Outlet</td><td>Liquid water</td><td rowspan="2">Pressure Outlet</td><td>Backflow temperature T_{out}^{W}</td></tr>
<tr><td>Refrigerant</td><td>Backflow temperature T_{out}^{R}</td></tr>
<tr><td rowspan="6">Walls</td><td>Glass</td><td>No slip condition</td><td>Heat Transfer Coefficient = 5 $\frac{W}{m^2\,K}$
Free stream temperature = 25.20 °C</td></tr>
<tr><td>Flanges</td><td rowspan="5">No slip condition

No slip condition</td><td rowspan="5">Adiabatic wall

Adiabatic wall</td></tr>
<tr><td>Refrigerant collector</td></tr>
<tr><td>Water collector</td></tr>
<tr><td>Seals</td></tr>
<tr><td>Nuts</td></tr>
</table>

1.4 Physical properties

The material properties used for CHT analysis are described in Table 4 for both fluids and solid parts. The assumption made in the study of the thermal analysis is shell conduction layer, were an oxide layer of 0.05 mm thickness and thermal conductivity of 0.05 W/m K is assumed between the water-copper-refrigerant interface.

Table 4: Material properties of fluids and solids

		Temperature [°C]	Density $\left[\frac{kg}{m^3}\right]$	Specific heat $\left[\frac{J}{kg\ K}\right]$	Thermal conductivity $\left[\frac{W}{m\ K}\right]$	Viscosity $\left[\frac{kg}{m\ s}\right]$
Fluid	Liquid water	20	998.2	4182	See Table 5.	See Table 5.
	Refrigerant	35	1311.2	1361.2	See Table 5.	See Table 5.
Solid	Copper	25	8978	381	387.6	-
	Glass	25	2230	830	1.2	-
	Brass	25	8730	380	109	-
	Stainless-steel	25	8000	502.4	14.4	-
	Teflon	25	2200	1090	0.25	-
	Polyaryl Ether Ketone (PAEK)	25	1320	1290	0.25	-
	Silicone	20	1500	1150	1	-

The polynomial functions [7] show the temperature-dependent properties including viscosity and thermal conductivity for both the fluids. In this way the properties have certain effect on the distribution of transient temperature fields during the simulation.

Table 5: Temperature dependent properties of the fluids

Liquid water	Viscosity	$y = -4.26073E{-}13T^5 + 7.10982E{-}10T^4 – 4.75468E{-}07T^3 + 1.59378E{-}04T^2 – 2.67992E{-}02T + 1.81067E{+}00$
	Thermal conductivity	$y = -6.59962E{-}12T^5 + 1.12648E{-}08T^4 – 7.65685E{-}06T^3 + 2.58100E{-}03T^2 – 4.28896E{-}01T + 2.85158E{+}01$
Refrigerant	Viscosity	$y = -1.74967E{-}14T^5 + 2.93208E{-}11T^4 – 1.98179E{-}08T^3 + 6.76906E{-}06T^2 – 1.17370E{-}03T + 8.33930E{-}02$
	Thermal conductivity	$y = -1.86703E{-}13T^5 + 2.71208E{-}10T^4 – 1.59512E{-}07T^3 + 4.76955E{-}05T^2 – 7.58849E{-}03T + 6.37220E{-}01$

1.5 Numerical method

CFD simulation using the ANSYS Workbench V18.1 has been performed to investigate the flow and heat transfer in the fluid and solid parts inside the measurement section. The ANSYS code solves the governing equations of the Navier-Stokes equations as well as the conjugate heat transfer using the finite volume method. Steady-state simulations have been performed with the SIMPLE algorithm. The Realizable k-e model with Enhanced Wall Treatment along with the first order upwind scheme and SIMPLE equation was used for solving turbulence and to calculate heat transfer for the fluid domain. For the stability of a solution the residual of 10E-6 has been maintained and monitoring points for velocity and pressure have been applied as a convergence criterion.

2 RESULTS AND DISCUSSION

Our numerical model has been validated with the temperature measurements in our previous publications [2] [3] [4] [5] [6]. There are two main goals of the present design optimization study: (a) reducing the heat loss of the system to the ambient air and (b) improving thermal isolation of both fluid flows inside the heads. As shown in the Figure 4, the measuring section has been divided into 3 different parts: inlet head, test section and outlet head. Visual observation of the flow boiling is possible through the transparent tube of the test section only. In order to achieve the largest heat flux in the test section, the fluid flows should be thermally isolated inside the heads. However, in reality there are always some heat losses present in the system. The actual heat losses in the present and optimized design have been extracted from our CFD results and are shown in Table 6. The heat fluxes have been calculated for each separate part of the measuring section shown in Table 6. Clearly, the optimized design has 50 % lower heat losses on both the heads and, instead, heat flux in the text section is increased by 16 %. Table 7 shows the heat transfer shares for the present and the optimized design. The total heat transfer is split to the three consecutive parts, i.e. inlet head, test section and outlet head. For the present design this ratio is 32:55:13 whereas for the optimized design they are 18:74:8 for inlet head, test section and outlet head, respectively. Clearly, the optimized design provides better thermal isolation of the heads and increased heat flux in the test section.

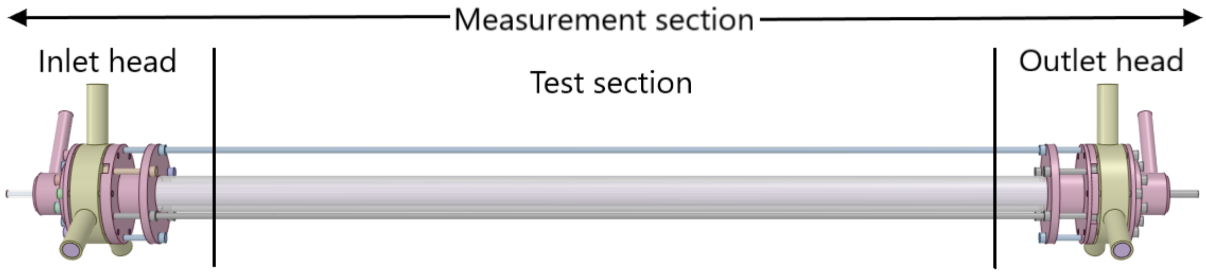


Figure 4: Test section division for heat loss calculation

Table 6: Heat loss at solid-fluid interfaces in Watts

	Present design		Optimized design	
Interfaces	Water-wall interface (W)	Refrigerant-wall interface (W)	Water-wall interface (W)	Refrigerant-wall interface (W)
Inlet head	-52.69	51.62	-26.38	26.06
Test-section	-89.28	90.45	-103.97	104.84
Outlet head	-21.08	20.94	-9.99	10.45

Table 7: Heat loss at solid-fluid interfaces in percentage

	Present design		Optimized design	
Interfaces	Copper-water interface (%)	Copper-Ref-Glass interface (%)	Copper-water interface (%)	Copper-Ref-Glass interface (%)
Inlet head	32	32	18	18
Test-section	55	55	74	74
Outlet head	13	13	8	8

2.1 Temperature field

Figure 5 shows the temperature distribution in the middle plain, which cuts the test section on two halves. The Figure 5 shows the temperature of hot water flow (red), cold refrigerant flow (dark blue) and all the internal parts of the inlet manifold, which extends into the test section. Water remains at a higher temperature much further in the optimized design (right) than in the present design (left), which means that the modification in the optimized design conducts less heat from the hot to the cold fluid inside the inlet head. A better insulation of both the fluid flows shifts the location with the largest heat flux from inside the inlet head downstream into the test section region. Here, a transparent outer pipe allows observation of the flow boiling at the hot copper surface.

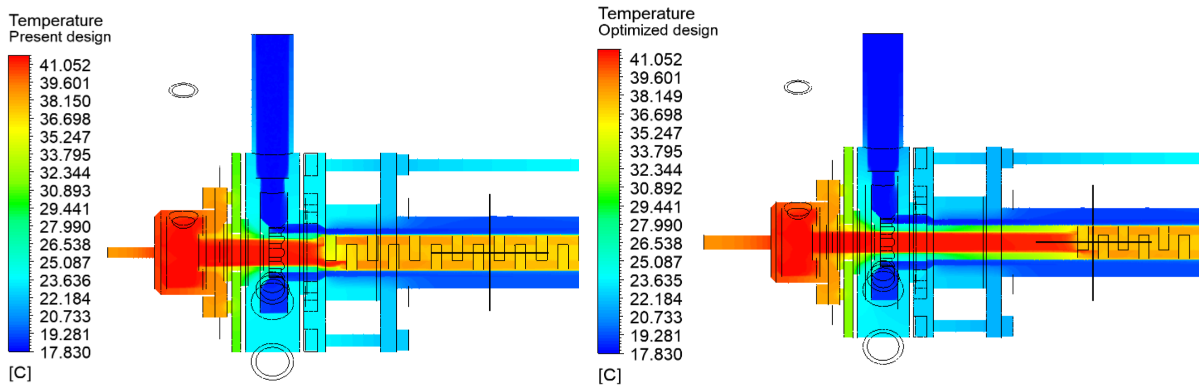


Figure 5: Temperature distribution of the present (left) and optimized design (right).

Figure 6 shows the temperature field at the refrigerant-solid interface of the refrigerant collector in the inlet head. The figure clearly shows that the highest temperature regions have been significantly reduced in the optimized design (right) with the respect to the present design (left).

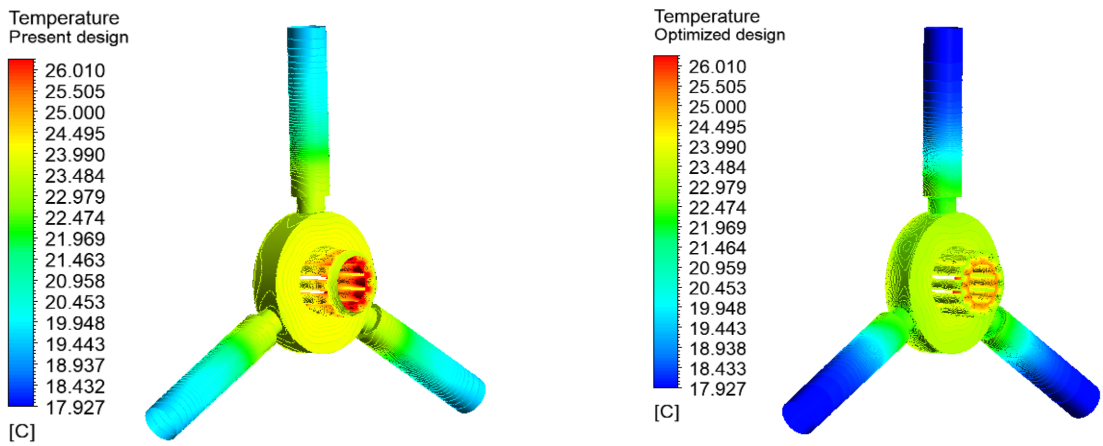


Figure 6: Temperature distribution at refrigerant collector-refrigerant interface

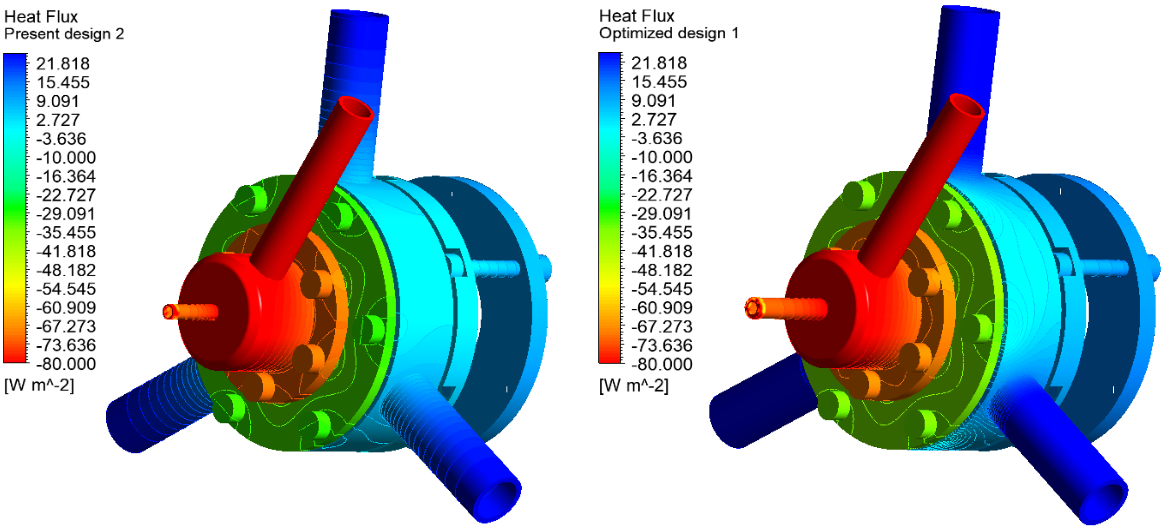


Figure 7: Heat flux on the surface of the head

The change in material of refrigerant collector from brass to stainless-steel also played an important role in limiting the heat flow to the ambient air. Figure 7 shows the heat flux on the surface of the head. It can be seen that the heat flux on the optimized design of refrigerant collector shows a reduced heat transfer with respect to the present design. This is attributed partially to the lower thermal conductivity of the refrigerant collector, which has been changed from brass to stainless steel. Part of the merits are attributed also to the inner tube inside the heads, which has been changed from stainless steel to plastic material and extended in length as we as in thickness.

3 CONCLUSION

Three-dimensional steady-state conjugate heat transfer model has been developed to investigate the heat transfer in the present and optimized design of the measuring section in the THELMA (Thermal-Hydraulics Experimental Laboratory for Multiphase Applications) experimental device. A CFD (Computational Fluid Dynamics) analysis allows a better insight into the flow and heat transfer of the measuring section, which is needed for proper determination of the total transferring heating power. The optimized design has showed 50 % better heat isolation between both fluid flows in the inlet/outlet heads. At the same time, about 15 % larger heat transfer is achieved in the test section for the simulated single-phase case. These results are very promising and even better performance is expected for two-phase boiling studies at larger heat fluxes.

ACKNOWLEDGMENTS

The authors would like to thank Dr. Marko Matkovič, Dr. Boštjan Končar and Dr. Mohit Pramod Sharma for useful discussions. The authors gratefully acknowledge financial support provided by Slovenian Research Agency, grants P2-0026, L2-9210 and L2-1827.

REFERENCES

[1] M. Omidi, M. Farhadi and M. Jafari, “A comprehensive review on double pipe heat exchangers”, Applied Thermal Engineering, vol. 110, pp. 1075-1090, 2017.

[2] M. Matkovič, L. Cizelj, I. Kljenak, B. Končar, M. Leskovar, A. Prošek and I. Tiselj, "Designing an advanced experimental test apparatus for conjugate heat transfer studies", SWINTH-16, Specialists Workshop on Advanced Instrumentation and Measurement Techniques for Nuclear Reactor Thermal Hydraulics, Livorno, June 15-17, 2016.

[3] M. Matkovič, L. Cizelj, I. Kljenak, B. Končar, B. Mikuž, A. Sušnik, I. Tiselj and B. Zajec, "Building a unique test section for local critical heat flux studies in light water reactor: like accident conditions", HEFAT2017, 13th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics, Portorož, Slovenia, July 17-19, 2017.

[4] A. K. Basavaraj, B. Zajec and M. Matkovič, "Accurate measurements of the local heat transfer coefficients along the dedicated test section," in 27th International Conference Nuclear Energy for New Europe - NENE2018, Portorož, Slovenia, September 10-13, 2018.

[5] A. K. Basavaraj, B. Mikuž and M. Matkovič, "Flow and Heat Transfer CFD Analysis in the Section of THELMA for Wall Surface Temperature Determination", 28th International Conference Nuclear Energy for New Europe - NENE2019, Portorož, Slovenia, September 9-12, 2019.

[6] B. Zajec, A. K. Basavaraj, B. Mikuž, Končar Boštjan, M. P. Sharma and M. Matkovič, "Experimental and Numerical Analysis of Heat Transfer on the Annular Test Section", 29th International Conference Nuclear Energy for New Europe - NENE2020, Portorož, Slovenia, September 7-10, 2020.

[7] P. J. Linstrom, W. G. Mallard, “NIST Chemistry WebBook, NIST Standard Reference Database Number 69”, National Institute of Standards and Technology, Gaithersburg MD, 2017.