Transactions of the VŠB – Technical University of Ostrava, Mechanical Series
No. 3, 2010, vol. LVI
article No. 1819

Petr KOVAŘÍK[*]

NUMERICAL SIMULATION OF FLOW IN THE ANNULAR TUBE

NUMERICKÁ SIMULACE PROUDĚNÍ V KANÁLU MEZIKRUHOVÉHO PRŮŘEZU

Abstract

This paper looks into numerical simulation of flow and heat transfer in the annular tube. For various values of inlet velocities are specified values of heat transfer in the annular tube and along the full length of the heated tube. The effect of various simplified spacer geometries on heat transfer in the annular tube is also monitored.

Abstrakt

Článek se zabývá numerickou simulací proudění a sdílení tepla v mezikruhového průřezu. Pro různou hodnotu velikosti vstupní rychlosti je určena velikost sdílení tepla v kanálu mezikruhového průřezu a podél celé topné tyče. Dále je sledován vliv různých zjednodušených geometrií distančního kroužku na sdílení tepla v mezikruhovém kanálu.

1 INTRODUCTION

Turbulent flow and heat exchange in heated annular tubes is a very interesting topic as it is present in many engineering systems, e.g. in heat exchangers, fuel rod clusters in nuclear reactors, etc. It is beneficial to understand the properties of turbulence and heat sharing in such flow, to create a model that describes these processes and, in turn, apply this model in development of such equipment. The numeric model of annular tube flow is being specified with the application of the measurements taken at the experimental apparatus that apply the hot-wire method (constant-temperature anemometry, CTA).

2 NUMERICAL SIMULATION

A geometry variant for 156-mm diameter turbulence generator with 36 open holes has been modeled, see Figure 2. Other models included variants with 36 open holes and simplified spacer geometry with lengths of 15 and 45 mm, at a distance from the heated tube wall of 1.5 and 0.5 mm, see Figure 1.

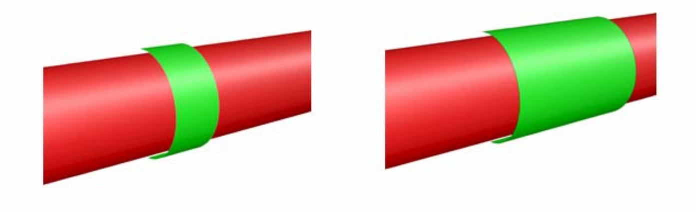

Fig. 1 Computational geometries: red – heated tube, green – spacer geometry

---

[*] Ing., Ph.D, University of West Bohemia, Department of Power System Engineering (KKE), Univerzitni 8, 306 14 Plzen, phone: +420 377 634 706, e-mail kovarikp@ntc.zcu.cz

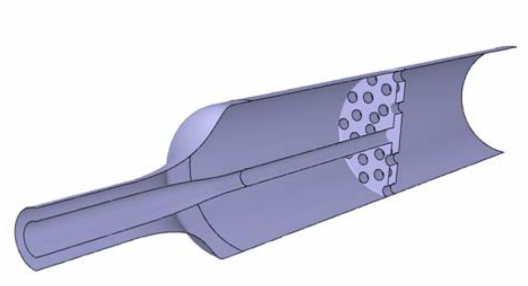

Fig. 2 Computational geometry – 36 holes

As the holes in the turbulence generator are symmetrical, the computational model was only used to represent one half of the true geometry and "symmetry" boundary conditions were applied to relevant walls. Only the inner part with the pipe was preserved from the inlet damper vessel. A "massflow – inlet" boundary condition was attached to the inlet of this pipe. For reasons of the applied mesh procedure, a boundary condition of "interface" is placed before the nozzle inlet behind the dampening section. A boundary condition "pressure – outlet" was attached to the outlet from the entire domain. Output generated by electricity passing through the heated tube was input as volume source into the volume of the heated tube. Air chambers inside the heated tube were modeled as "solid" with thermophysical properties of the air. This prevents air mixing inside the heated tube induced by heating. The computational mesh of the entire domain contained approximately 5 million cells, channel height was 60 cells. Computational geometry of the symmetrical fuel cluster model was prepared using the Gambit 2.4.6 software package. The numerical simulation was run in Fluent 6, a commercial CFD software package 6.3.26.

3 RESULT

60 and 15 m/s inlet velocities in the annular tube were selected for all of the geometry variants. Heating output corresponded to passing electric current of 150 A.

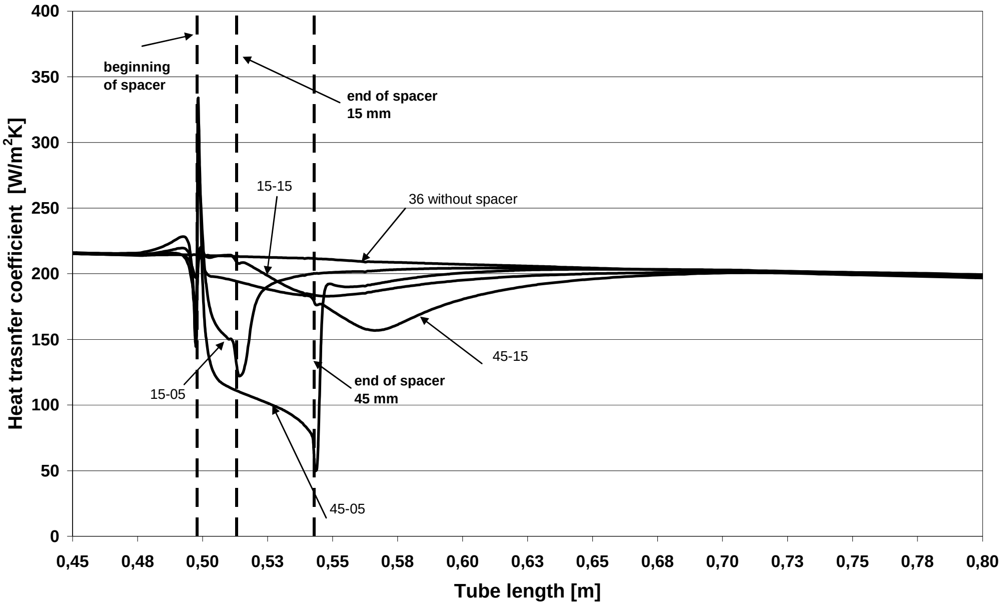

Fig. 3 Development of the heat transfer coefficient – 60 m/s velocity

Figures 3 and 4 show the development of heat transfer coefficient along the heated tube. For the lower velocity variant, the heat transfer coefficient is approximately three times lower. From the development point of view, the largest difference was identified in the variants that are located at 1.5 mm from the heated tube where the lower velocity variants showed increment of the coefficient behind the spacer end. For the variants with a distance of 0.5 mm from the wall, more rapid increment of the coefficient occurs behind the spacer.

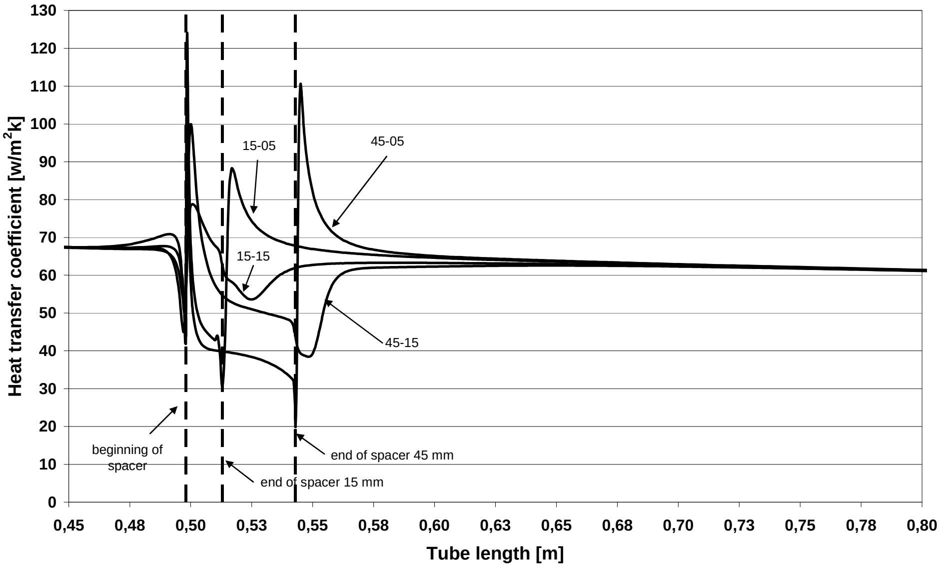

Fig. 4 Development of the heat transfer coefficient – 15 m/s velocity

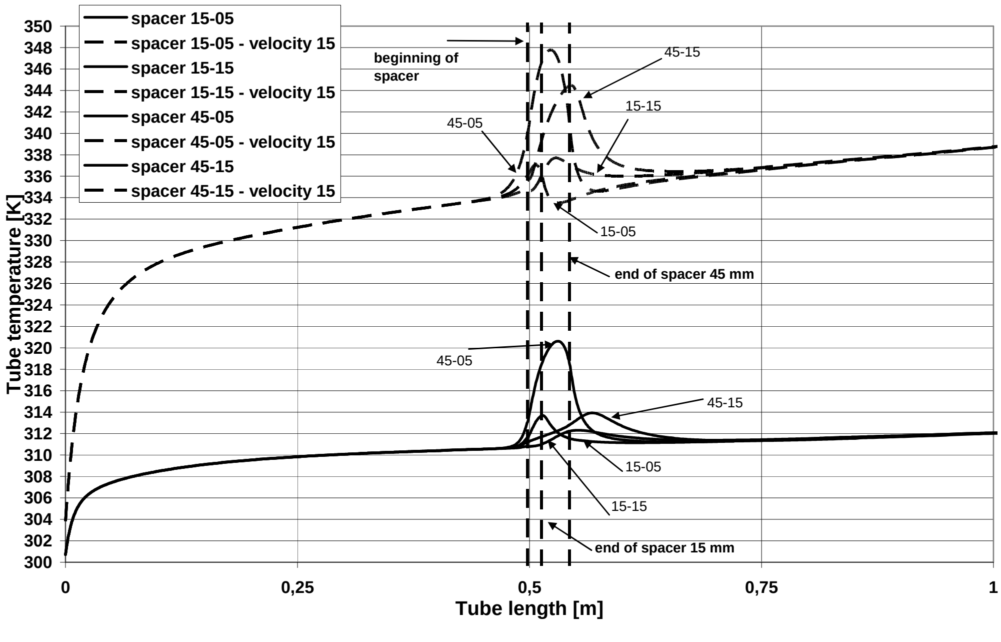

Fig. 5 Temperature development along the heated tube

Figure 5 shows temperature development along the heated tube (solid line for 60 m/s velocity, broken line for 15 m/s velocity). Reduction of the inlet velocity in the annular tube to ¼ caused heated tube temperature to increase by approximately 26 degrees. Due to the presence of the spacer, heated tube temperature is higher in the location of the spacer. For velocity 60 m/s, the heated tube reports lowest temperature increment with the 15 mm long spacer that is located at 1.5 mm from the tube; for velocity 15 m/s this is seen in the variant with the spacer length of 15 mm and distance from the tube of 0.5 mm. Highest temperature increment, for both velocities, is seen at the spacer variant with spacer length of 45 mm, located at 0.5 mm from the wall. This is caused by velocity reduction in the gap between the heated rod and spacer.

In the variants with 15 m/s velocity, another shift also occurs in the location of the maximum temperature spot which is then found closer to the end of the spacer. Furthermore, faster cooling of the heated tube occurs behind the spacer.

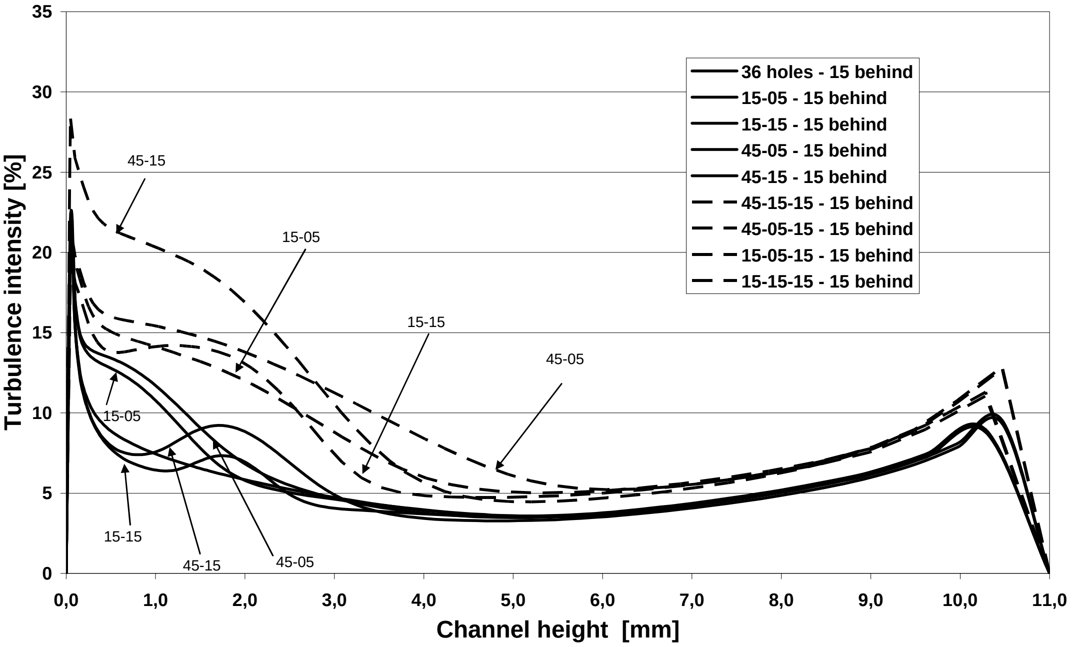

Fig. 6 Turbulence intensity development at channel height – 15 mm behind the spacer

Development of turbulence intensity at 15 mm behind the spacer is shown in Figure 6. It is clear that for the velocity of 60 m/s, turbulence profiles are identical as early as at 4 mm from the tube wall. Turbulent profile becomes balanced at approximately 250 mm behind the spacer end. For the velocity of 15 m/s, the profiles along the channel height are not straightened until at 6 mm from the tube wall. Higher turbulence in the left section of the chart is, in a certain extent, caused by the definition of turbulence intensity examination [3], where the turbulent kinetic energy is divided by local velocity value; therefore, turbulence intensity is higher near the walls.

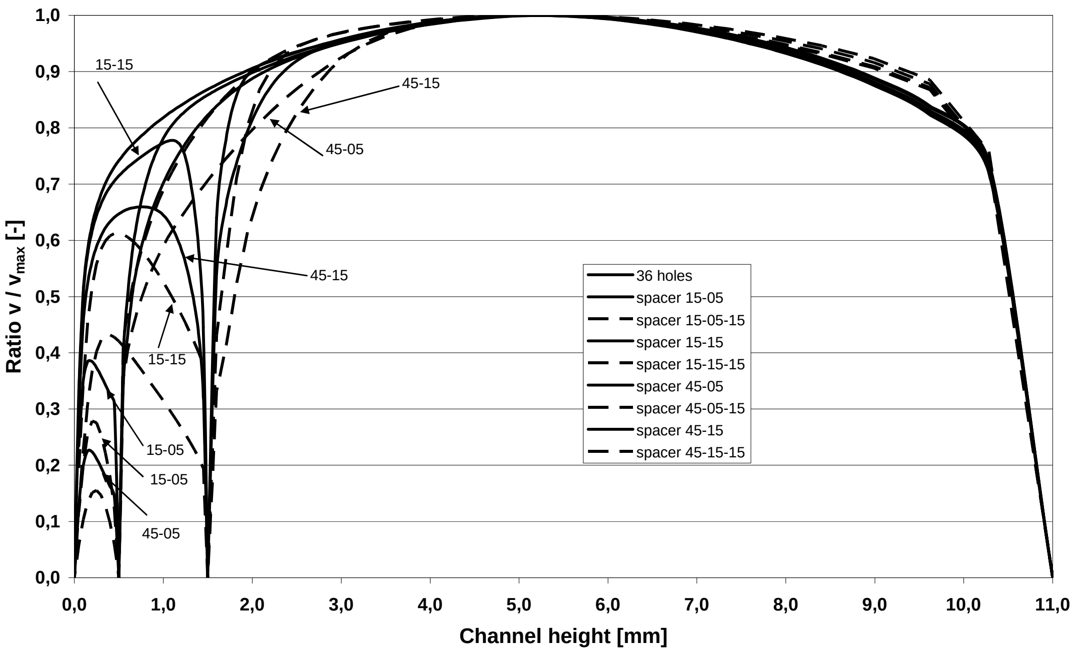

Fig. 7 Velocity development along channel height: centre of spacer

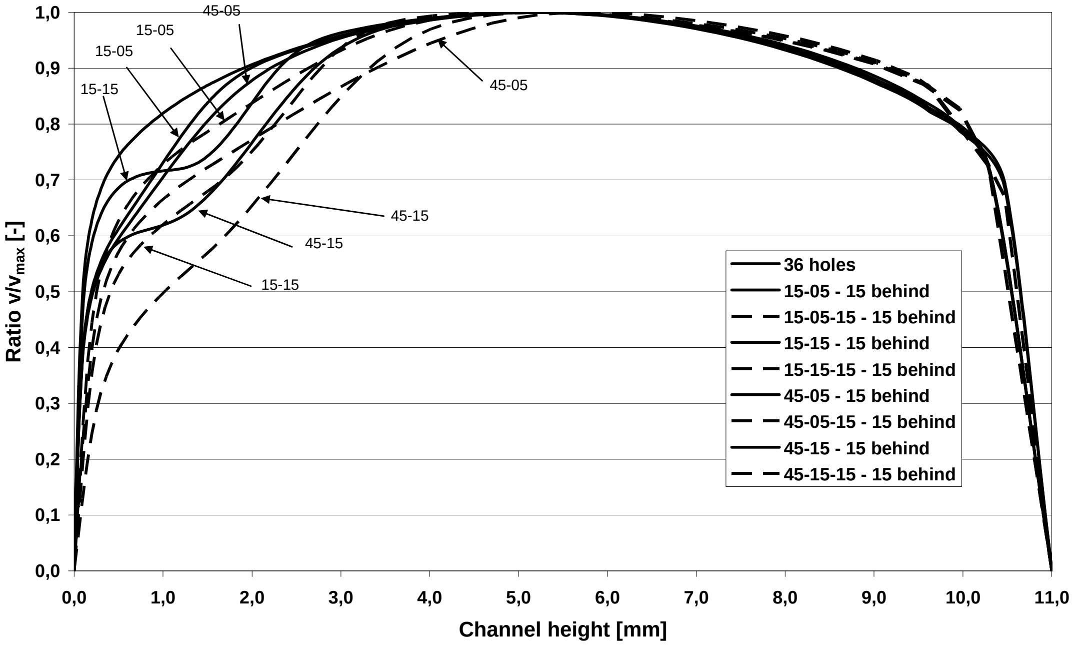

Fig. 8 Velocity development along channel height: 15 mm behind spacer

Velocity development by channel height is shown in the charts of Figure 7 and Figure 8. The chart on the Figure 7 depicts the plane at one half of spacer length. The variant with spacer length of 15 mm and 1.5 mm distance from the heated rod shows development identical to that of the basic variant up to ca. 0.5 mm from the wall; the effect of the spacer is seen in larger distances.

The most distinctive velocity reduction in the area between the spacer and heated tube occurs at the spacer variant with length of 45 mm and distance 0.5 mm from the wall. After approximately 3 mm from the wall, the velocity profile is identical for the velocity of 60 m/s, regardless of whether the spacer is present or not. The following chart (Figure 8) shows velocity profiles at 15 mm behind the end of the spacer. The chart clearly indicates profiles to be more balanced as the distance from the spacer end increases. With the velocity of 60 m/s, its profile becomes balanced at a distance of approximately 350 mm behind the spacer. For the velocity of 15 m/s, the balance occurs at approximately 400 mm.

4 CONCLUSIONS

Numerical simulations of air flow and heat transfer in the annular channel have been performed. The results of the simulation have indicated the effects of simplified spacer geometry and reduction of inlet velocity to the velocity and temperature fields. With speed reduced to ¼, the coefficient of heat transfer reduces approximately three times while temperature rises by about 26 degrees. In terms of flow field disruption, the smallest effects have been demonstrated at the shortest spacer with the longest gap. Reduced velocities also show more prevalent effect of the spacer being present in the annular tube. In terms of heat transfer, the largest reduction of heat transfer coefficient occurs in the layout using the longest spacer with the smallest gap. It has been demonstrated that the temperature of the heated rod increases where the spacer is located. As there are several spacers used in the real rod, assumptions may be made towards potential deformation of the heating tube by the higher difference in temperature.

Acknowledgement

The making of this work was supported with funds from the Grant Agency of the Czech Republic as part of post-doctoral project 101/08/P331.

REFERENCES

[1] LINHART J.: Vliv vstupní turbulence na proudění a sdílení tepla v kanálu mezikruhového průřezu. *Výzkumná zpráva*, Místo: VŠSE, 1972

[2] KLÁŠTERKA H.: Vliv kroužku na proudění a sdílení tepla v mezikruhovém kanálu, *Výzkumná zpráva*, Místo: VŠSE, 1974

[3] KOVAŘÍK P.: Numerická simulace sdílení tepla v kanálu mezidruhového průřezu.In *XVII. Medzinárodná vedecká konferencie Aplikácia experimentálnych a nemurických metód v mechanike tekutín a energetike.* s. 137 – 142. ISBN 978-80-554-0189-8

[4] KANG S., PATIL B., ZARATE, ROY R.P: Isothermal and heated turbulent upflow in a vertical annular channel - Part II. Numerical simulations , *In Journal of Heat and Mass transfer Vol. 44*, s. 1185-1199, 2001