Simulation of turbulent heat transfer in annular flow with asymmetric wall roughness
G. H. Klepp, G. H. D. Mühlenbruch & J. Schmidt
Institute of Fluid Dynamics and Thermodynamics, Otto-von-Guericke University, P.O. Box 4120, D-39016 Magdeburg, Germany
EMail: georg.klepp@student.uni-magdeburg.de

Abstract

The heat transfer in stationary, fully developed annular flow with static or moving cores and asymmetric wall roughness is simulated. In order to describe the fully developed flow field a Reynolds stress model for high Reynolds numbers is derived in the cylindrical coordinate system. The accurate description of the highly asymmetric annular flow is a prerequisite for the heat transfer computation, the turbulence transport properties being known, the heat transfer may be readily computed. The near wall region is described by an appropriate distribution law and using the Boussinesq approach. Temperature and flow fields are computed on two different grids, using a finite difference scheme on stretched grids.

The model is validated by comparison with experimental single phase convective heat transfer and pressure loss data for annuli with artificial roughness at the outer wall. For the intensive cooling of hot-rolled wires in cooling tubes with turbulence promoters at the outer, adiabatic wall extensive experimental data is available. The cooling of hot rolled wires with a smooth surface in the film boiling region is modelled, using an equivalent roughness for the turbulence promoters.

1 Introduction

A well-known method to enhance the heat transfer is to roughen the heat transfer surface. For certain industrial applications, as the cooling of hot rolled rods in cooling tubes, artificial roughness elements on the wall of
the tubes are used to increase the heat transfer on the surface of the rolling stock. The hot rolled rods enter the cooling tube after the last roll stand and are quenched by water in co-current flow, the turbulence and thus the heat transfer being enhanced by turbulence bushes at the outer wall of the tube (compare Fig.1). The annular flow in these intensive cooling tubes is strongly asymmetric due to the moved core and the asymmetric wall roughness.

Most studies concerning heat transfer in asymmetric turbulent flow use simplified models and the results are only applicable to certain types of flow, e.g. Shigechi et al. [1] and Torii and Yang [2], or adjustments have to be made, Barrow and Pope [3]. In addition these models show some differences in their dependency from the main influence parameters. Due to a lack of sufficient experimental data an assessment of those models is difficult. Also the simulation of the heat transfer in intensive cooling tubes as presented in [4] is restricted to this particular industrial application. Thus a general model to describe heat and momentum transfer in asymmetric turbulent annular flow on a wider physical basis would be useful.

2 Model

2.1 Flow model

The flow in the annulus with a moving core and artificial roughness elements on the inside of the outer wall is described by the Reynolds-averaged equation of continuity

$$\nabla U = 0$$

(1)

and the Reynolds-averaged Navier-Stokes equations

$$\frac{D U}{D t} = - \frac{1}{\rho} \nabla P + \nu \Delta U$$

(2)
of an incompressible fluid, which are simplified by assuming constant fluid properties and neglecting additional forces. In order to simplify the model, the following additional assumptions were made.

- The annulus is concentric.
- The flow is stationary, irrotational and fully developed.
- The influence of artificial roughness elements is considered by a hydraulic diameter and an equivalent roughness.
- The flow can be divided into a region of fully developed turbulence and two near wall regions.
- The Reynolds stress model is independent of the molecular viscosity, as it describes only the region of the fully developed turbulent flow.

Confining the model to the fully developed turbulent flow in a concentric annulus, simplifies the system of governing equations. Time-averaging and subsequent integration of the Navier-Stokes equations in the axial direction gives an analytical relation for the only Reynolds shear stress component to be considered

$$u'v'(r) = \frac{r_a u_i v_i - r_i u'_i v'_i}{r_a^2 - r_i^2} \frac{r_a^2 u_i v_i - r_i u'_i v'_i}{r_a^2 - r_i^2} \frac{r}{r_i}.$$  (3)

The Reynolds transport equations are deduced in cylindrical coordinates, similar as in Launder, Reece and Rodi [5], leading to the transport equation for the turbulent kinetic energy.

$$0 = - \frac{u'v'}{u'v'} \frac{du}{dr} - \varepsilon + \frac{c_s}{r} \frac{d}{dr} \left( r \frac{k}{\varepsilon} \left( c_v k \frac{dk}{dr} + c_w k \frac{dk}{dr} + u'v' \frac{du'v'}{dr} \right) \right)$$

$$+ \frac{c_s}{r} \frac{r}{\varepsilon} \frac{d}{dr} \left( \frac{k^3}{\varepsilon} \right)$$  (4)

The additional diffusion-term results from deducing in cylindrical coordinates. The same procedure leads to the transport equation for the Reynolds shear stress.

$$0 = - c_s \frac{u'v'}{k} \frac{\varepsilon}{k} + (c_v^2 - 1) c_v k \frac{du}{dr} + \frac{2 c_s c_v}{r} \frac{du}{dr} \left( r \frac{k^2 du'v'}{dr} \right)$$

$$- c_s \frac{k}{\varepsilon} \frac{u'v'}{r} \frac{dk}{dr} + 2 \frac{k}{r}$$  (5)
Advanced Computational Methods in Heat Transfer

The simplified transport equation for the turbulent dissipation, as cited in the literature [5], was often applied successfully. The direct transformation in cylindrical coordinates yields

\[ 0 = -c_{e1} u^r \frac{E}{k} \frac{du}{dr} - c_{e2} \frac{E^2}{r} \frac{d}{dr} \left( r \frac{k^2 dE}{dr} \right) \]

(6)

The model constants, which describe the ratio of the velocity fluctuations to the turbulent kinetic energy, are described as parabolic functions

\[ c_j = c_j^0 + c_j^w \left( \frac{2 \tau - (r_{mi} + r_{ma})}{r_{ma} - r_{mi}} \right)^2 \]

\[ j = v, w \]

(7)

<table>
<thead>
<tr>
<th>constant</th>
<th>value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( c_1 )</td>
<td>0.11</td>
</tr>
<tr>
<td>( c_{e1} )</td>
<td>1.5</td>
</tr>
<tr>
<td>( c_{e2} )</td>
<td>0.6</td>
</tr>
<tr>
<td>( c_2 )</td>
<td>0.20</td>
</tr>
<tr>
<td>( c_4 )</td>
<td>1.44</td>
</tr>
<tr>
<td>( c_5 )</td>
<td>1.92</td>
</tr>
<tr>
<td>( c_v^0 / c_w^0 )</td>
<td>0.487 / 0.547</td>
</tr>
<tr>
<td>( c_v^w / c_w^w )</td>
<td>0.24 / 0.03</td>
</tr>
</tbody>
</table>

Table 1: Model constants

as proposed by Eder [6] in order to consider the influence of the walls.

All model constants used are shown in Table 1. They were validated against experimental data from the literature, [9], and are similar to the values used by other investigators (Lauder [5]).

Based on the logarithmic velocity distribution law according to Eder [6], the velocity in the near wall region is described by the equation

\[ u = u_w^* + \frac{1}{\kappa} \ln \left( 1 + \kappa \frac{x}{y^+} \right) + a_u \left( 1 - e^{-y^+ / y_1^+} - \frac{y^+}{y_1^+} e^{-0.3 y^+} \right) \]

(8)

with

\[ a_u = \frac{1}{\kappa} \ln \left( \frac{e^{\kappa B_G}}{\kappa^{1.04}} \right) \quad \text{and} \quad \frac{\kappa}{\kappa B_G} = \left( 1 + \frac{k_w^+}{e^{(B_R - B_G)}} \right)^{-1} \]

The parameter \( y_1^+ \) describes the thickness of the viscous sublayer. At the inner wall, the velocity profile is influenced by the ratio of the wall radii. The use the transformed coordinate \( y^+_r = r^*_i \ln \left( r/r_i \right) \) permits the description by the same equation.

2.2 Roughness modelling

The artificial roughness elements on the outer wall are described by two parameters: the hydraulic diameter \( d_{hy} \) (see Fig. 1) and the equivalent roughness \( k_w \), which has to be determined using experimental data.
dependence on the type of roughness elements. The choice of these two parameters distinguishes this model from other similar models (e.g. [7]) which introduce an additional term in dependence on the geometry and on the flow conditions. In an earlier approach to model the heat transfer in cooling tubes [4], simplifying assumptions concerning the hydraulic diameter were introduced and thus it was not possible to reproduce both the experimental pressure drop as well as the experimental heat transfer data with good accuracy.

Therefore, an effort was undertaken to get a good agreement of the computed pressure drops with measured values in single-phase flow. The best agreement with experimental results, [9], is reached with a hydraulic diameter based on the volume of a plane gap with the same shape

\[ \frac{d_{h,B}}{d_{\text{clear}}} = 1 + 0.3 \frac{d(V_{\text{plane}})}{d_{\text{clear}}} \]  

(9)

and the equivalent roughness defined by

\[ k_{w,B} = 0.22 \left( d_{\text{max},B} - d_{\text{clear}} \right) . \]  

(10)

With these model parameters, the influence of the flow rate and the inner diameter, including gap width and the ratio of the roughness to the equivalent diameter of the annulus, can be reproduced by the model (Fig. 2). Using these model parameters for the case of the annular flow with moving core, the good agreement between the measured and the computed values enabled by the use of the Reynolds stress model. Introducing the slip model permits the consideration of the small influence of the vapour phase (Fig. 3).

![Figure 2: Comparison of numerical to experimental pressure loss per length for different annular gaps (\(u_w = 0 \text{ m/s}\)).](image1)

![Figure 3: Comparison of numerical to experimental pressure loss per length for cooling of hot rolled steels.](image2)
3 Results and comparison with experimental data

3.1 Single-phase flow

In Fig. 4 the influence of the roughness of the adiabatic outer wall on the heat transfer at the inner wall is shown. The correlation of Petukhov and Roizen [10] for the case of smooth outer wall is reproduced with very good accuracy.

The Nusselt number (Nu) increases with increasing Reynolds numbers (Re) and increasing relative roughness heights (k_{wa}/d_{equ}). The increase of the Nusselt-number for the range of equivalent relative roughness heights occurring in the intensive cooling tube is computed and reproduces the corresponding experimental data of Kühne [12] with a deviation of about 10%. Using artificial roughness elements at the adiabatic outer wall of the intensive cooling tube may increase the heat transfer to up to 15%.

3.2 Film boiling in intensive cooling tubes

The model is used to investigate the influence of artificial roughness elements for "convection controlled" film boiling in the intensive cooling tube for Couette-type flow. For this case the influence of the moving core as well as of the two-phase flow and heat transfer in the near wall region have to be taken into consideration. Numerical and measured data in the range of relative roughness heights used in the investigated intensive cooling tube are presented in Fig.5.

For the “convection controlled“ film boiling region the heat transfer coefficients are significantly higher then those for single phase convection. The increase in roughness at the outer wall of the annulus
also slightly increases the heat transfer. With an increase in the Reynolds number the Nusselt numbers decrease for the annuli with smooth walls. For the roughened annuli this dependence is not so distinct and for the greatest roughness considered in this investigation the Nusselt number increases with increasing Reynolds numbers.

Figure 5: Influence of the outer wall roughness on heat transfer in the region of "convection controlled" film boiling.

With an increase in the fluid velocity the difference to the velocity of the moving rod decreases and the vapour film thickness as well as the thickness of the thermal boundary layer increases reducing the wall shear stress and the heat transfer. This effect leads to the decrease in Nusselt numbers with increasing Reynolds numbers in the smooth annuli. The increasing fluid velocity also increases the turbulent transport. Thus the reduction in heat transfer due to decreasing Reynolds numbers in Couette type flows is not so significant for rough surfaces and for the highest relative roughness $k_{wb}/d_{equ} = 0.068$ the Nusselt number even increases.

4 Conclusions

A model to describe momentum and heat transfer in fully developed annular flow with a moving core and asymmetric wall roughness was presented. Due to the general approach based on an appropriate turbulence modelling, this model may be used to describe different types of asymmetric turbulent annular flow.

The model is validated using experimental heat transfer data for single phase convection with asymmetric wall roughness. Film boiling heat transfer of hot rolled rods in intensive cooling tubes is modelled using this one-phase model which is adjusted to describe two-phase flow phenomena. The measured data is reproduced with satisfying accuracy. Thus this heat transfer simulation should reduce the experimental cost for the design of cooling lines in mill trains and for the design of new...
types of cooling tubes. Using the presented model, also other practical applications may be simulated: the heat transfer from gas cooled nuclear fuel rods in high temperature reactors or the flow and heat transfer in railway tunnels.

References


Numerical modelling of phase ‘A’ of thermal explosions occurring in different conditions

T. R. Fodemski & A. Rosiak

Technical University of Łódź
Institute of Heat Technology and Refrigeration
ul. Stefanowskiego 12/16
90-924 Łódź, Poland

Abstract

This work was undertaken as part of an investigation of a thermal explosion phenomenon, occurring occasionally under specific conditions, when two liquids of different volatility, are mixed. The ‘thermal explosion efficiency coefficient’ is used for the determination of the fraction of the thermal energy initially available in the hot liquid (less volatile, called ‘fuel’) which is converted into destructive mechanical energy by the explosive expansion of the colder liquid (volatile, called ‘coolant’). This complex phenomenon consists of separate stages, forming a chain of many events, which must occur – if an effective and large scale thermal explosion is to take place. Two separate phases ‘A’ and ‘B’ are usually distinguished. In phase ‘A’ a very fast heat transfer, between fragmented fuel particles and liquid coolant, occurs. Since the scale of the fuel fragmentation is usually very fine and both liquids are in direct contact, significant amount of the thermal energy, during this phase ‘A’ duration time scale of order of $10^{-3}$ s, is transferred into the coolant. This paper is concerned with the presentation of the numerical calculation method, using Mathcad Plus 6.0, of the following time histories: temperature and heat flux on the fuel/coolant interface and, also, temperature profiles in the fuel and in the coolant. All these quantities are crucial for the accurate estimation of the thermal explosion efficiency coefficient. The method is applied to different pairs of fuel and coolant liquids – characteristic of possible thermal explosion in the nuclear, metallurgical and liquefied gas industries. The influences of the following important parameters (related to phase ‘A’ of this phenomenon), such as size and shape of fuel particles, the coolant/fuel mass ratio and the initial fuel and coolant temperatures, for each liquid pair, are analysed.