Annals of Nuclear Energy 117 (2018) 32–44

Contents lists available at ScienceDirect

Annals of Nuclear Energy

journal homepage: www.elsevier.com/locate/anucene

Application of an algebraic turbulent heat flux model to a backward facing step flow at low Prandtl number

Andrea De Santis, Afaque Shams *

Nuclear Research and Consultancy Group NRG, Westerduinweg 3, 1755 LE Petten, The Netherlands

ARTICLE INFO

Article history:
Received 16 November 2017
Received in revised form 6 March 2018
Accepted 8 March 2018

Keywords:
Turbulent heat transfer
Low Prandtl fluids
Algebraic heat flux model
Forced convection
Mixed convection

ABSTRACT

Turbulent heat transfer represents a considerably challenging phenomenon from the modelling point of view. In the RANS framework, the classical Reynolds analogy provides a simple and robust approach which is widely employed for the closure of the turbulent heat flux term in a broad range of applications. At the same time, there is an ever growing interest in the development and assessment of advanced models which would allow, at least to some extent, for the relaxation of the simplifying assumptions underlying the Reynolds analogy. In this respect, the use of algebraic closures for the turbulent heat flux has been proposed in the literature by different authors as a viable approach. One of these algebraic closures has been extended for its application to low Prandtl number fluids in various flow regimes, by means of calibration and assessment of the model against some basic test cases, in what is known as the AHFM-NRG+ model. In the present work the AHFM-NRG+ is applied for the first time to a relatively complex configuration, i.e. a backward facing step in both forced and mixed convection regimes with a low Prandtl working fluid, and assessed against reference DNS data. The obtained results suggest that the AHFM-NRG+ is able to provide more accurate predictions for the thermal field within the domain and for the heat transfer at the wall in comparison to the Reynolds analogy assumption. These encouraging results indicate that the AHFM-NRG+ can be considered as a promising model to improve the accuracy in the simulation of the turbulent heat transfer in industrial applications involving low Prandtl fluids.

1. Introduction

Due to their high thermal conductivity, liquid metals represent an attractive option as a coolant fluid. In fact, such fluids are currently considered in a broad range of industrial applications including the production of steel (Felten et al., 2004) and semiconductors (Raufeisen et al., 2009), and in thermal solar plants (Frazer et al., 2014). With respect to nuclear fission applications, liquid metals are foreseen as the primary coolant fluid in two of the six concepts considered within the Generation IV International Forum (GIF), i.e. the Lead Fast Reactor (LFR) and the Sodium Fast Reactor (SFR) (GIF, 2010). In this context, thermal–hydraulics is regarded as one of the key issues in the development of liquid metal cooled reactors. The complexity of the thermal–hydraulics in such reactors stems from a number of factors, including (Roelofs et al., 2015):

- Complex flow field with significant turbulence anisotropy.
- Non-negligible buoyancy influence.
- Low Prandtl number (in the order of 0.01–0.001) of the working fluid.

Computational Fluid Dynamics (CFD) is regarded as a tool of great importance in order to overcome the aforementioned issues (Grötzbach, 2013). In the framework of the RANS approach and in light of the points stressed above, it is worth to point out that a broad range of models have been developed in the past decades with respect to the turbulent momentum transfer (Shams, 2017a). This allows for the choice of suitable models to account for turbulence anisotropy (e.g. Reynolds Stress Models (Hanjalić, 1999) or non-linear Eddy Viscosity Models (Bauer et al., 2000)). Therefore, at least in the forced convection flow regime, the uncertainties associated with the modelling of the turbulent flow field can be minimised by resorting to an appropriate turbulence model.

Furthermore, most turbulence models available in CFD codes also feature dedicated source terms in the momentum and in the turbulence equations in order to account, at least partially, for buoyancy effects (Grötzbach, 2013). Nevertheless, when buoyancy

* Corresponding author.
E-mail address: shams@nrg.eu (A. Shams).

https://doi.org/10.1016/j.anucene.2018.03.016
0306-4549/

has a strong influence on the flow, the temperature cannot be considered as a passive scalar since it can be expected to have a significant impact on the flow field. As a consequence, an accurate evaluation of the thermal field is of significant importance in such cases (Grötzbach, 2013).

Unfortunately, differently from the turbulent momentum transport, only a limited number of models are available for the closure of the turbulent heat flux (THF) term (Grötzbach, 2007). Very often the only option available in commercial CFD codes relies on a Simple Gradient Diffusion Hypothesis (SGDH), in which the THF is assumed to be proportional to the mean temperature gradient through the turbulent thermal diffusivity α_t. Furthermore, in order to evaluate α_t, the so called Reynolds analogy is almost universally invoked. Under this hypothesis, similarity in the turbulent transport of momentum and heat is assumed, and the turbulent thermal diffusivity is evaluated from the momentum diffusivity by introducing the turbulent Prandtl number Pr_t. The assumption of similarity between turbulent momentum and heat transport allows to simplify the problem significantly, but it can result in an oversimplification with respect to the actual physics of the flow in some configurations (Roelofs et al., 2015). In addition, in most cases Pr_t is assumed to be a constant (usually having a value around 0.9–1.0), although broad experimental and numerical evidence has shown that this is a major simplification with respect to the actual physics (Grötzbach, 2013). Despite this substantial simplifying assumptions, the Reynolds analogy can represent a reasonable compromise for a broad range of applications and has been the workhorse for RANS turbulent heat transfer modelling for decades. Nevertheless, the nuclear community is well aware of its possible shortcomings when considering complex applications as those involving low Prandtl fluids and/or buoyant flows. As a consequence, in the European project THINS (Thermal–Hydraulics of Innovative Nuclear Systems), more advanced closures for the THF have been proposed in order to improve the accuracy of the CFD models dealing with the turbulent heat transfer at low Prandtl number (Shams, 2017a). The models considered within this project include a local turbulent Prandtl number model (Kays, 1994; Duponcheel et al., 2014), and both an explicit (Manservisi and Menghini, 2014) and an implicit Algebraic Heat Flux Model (AHFM) (Kenjereš et al., 2005; Shams et al., 2014). The general conclusion within the THINS project was that the efforts towards the development of new THF models should be limited, and the focus should be put on further validation of the proposed models in more complex configurations (Shams, 2017a).

With respect to the implicit AHFM approach, the model originally proposed in Kenjereš et al. (2005) for natural and mixed convection with unity Prandtl number fluids has been extended for the application to forced convection and low Prandtl fluids in the so-called AHFM-NRG model (Shams et al., 2014). The calibration and assessment of the model have been performed against a set of simple test cases (i.e. forced planar and wavy channel flows, Rayleigh-Bnard convection, vertical heated channel in mixed convection). In addition, this model has been further extended to natural convection at high Rayleigh number Ra in Shams (2017b), resulting in the so called AHFM-NRG+ model.

One of the main hampering factors with respect to the further assessment of the proposed THF models in complex configurations is the lack of reference data, both experimental and numerical, for their validation. Therefore, the two European projects SESAME and MYRTE, which are currently ongoing, have the aim of generating such a reference database in order to be able to further assess the proposed THF models against comprehensive validation data (Shams, 2017a).

In this context, the present work aims at contribute to the further understanding of the performance of different THF closures in a relatively complex case that is relevant for nuclear applications. In particular the classical Reynolds analogy and the AHFM-NRG+ model have been employed to simulate the turbulent heat transfer in a backward facing step (BFS) flow in both forced and mixed convection with a low Prandtl working fluid. The reference database is a recently published DNS reported in Niemann and Fröhlich (2016). A description of the computational domain and of the numerical settings is reported in Section 2, followed by an overview of the turbulence models adopted in the present RANS calculations in Section 3. Successively, the results obtained for both the forced and the mixed convection cases are illustrated and compared against the reference DNS data in Section 4. Finally, conclusive remarks along with future perspectives are provided in Section 5.

2. Computational domain and simulation settings

The computational domain, which has been employed for both the forced and the mixed convection calculations, reproduces the reference DNS domain Niemann and Fröhlich (2016) and is sketched in Fig. 1. The domain represents a vertically oriented BFS geometry with a step height equal to h and an expansion ratio of 1.5. In contrast with the reference DNS, in which a periodic condition is imposed in the spanwise z direction, the computational domain considered in this work is two-dimensional.

In terms of boundary conditions, similarly to what has been done in the reference DNS, a fully developed inflow condition has been generated through a periodic simulation of an inlet channel. The velocity and turbulence profiles obtained from the periodic calculation have been imposed at the BFS inlet. For the temperature, a uniform inlet value of 423.15 K has been used. At the outlet section, a Dirichlet condition has been used for the pressure, whilst a homogeneous Neumann condition has been adopted for all the other variables. All the remaining boundaries have been treated as adiabatic no-slip walls, with the exception of the bottom wall downstream of the step, where a heat flux of 41 kW/m^2 has been imposed for $x < 20h$.

The working fluid is liquid sodium. The changes in the physical properties of the sodium with respect to the temperature have been neglected, and all the properties have been evaluated from Sobolev (2011) at the reference temperature T_{ref}, which has been taken equal to the inlet temperature of 423.15 K. At this temperature the Prandtl number of the sodium is equal to 0.0088. In addition to the Prandtl number, the other non-dimensional groups defining the problem are the Reynolds number $Re = 2hU_b/\nu$ and the Richardson number $Ri = g\beta\Delta Th/U_b^2$. The Reynolds number is calculated with respect to the inlet channel width $2h$ and the bulk inlet velocity U_b and is equal to 9610. The temperature difference ΔT is defined with respect to the imposed heat flux q'' as $\Delta T = q''h/\lambda$, where λ is the thermal conductivity of the sodium, and is equal to 20 K. In the DNS reference data, two different values of the Richardson number have been considered. The first one is $Ri = 0$, which is enforced by setting the thermal expansion coefficient β to zero, and corresponds to the forced convection regime. The second Ri value, which is equal to 0.338, corresponds to the mixed convection regime. Both cases have been modelled in the present calculations.

All the simulations have been performed using the commercially available software STAR-CCM+ version 11.06 (SIEMENS PLM Software, 2016). It is worth to point out that the considered AHFM model has been implemented within this code in the framework of the THINS project. The governing equations have been solved in their steady-state formulation and the SIMPLE algorithm has been used for pressure–velocity coupling (Patankar and Spalding, 1972). A second order scheme has been employed for the spatial discretisation of all the equations.