Turbulence Model Comparison for Compact Plate Heat Exchanger Design Application.
F. Vitillo, L. Cachon, P. Millan, P. Reulet, E. Laroche

To cite this version:

HAL Id: hal-01059010
https://hal-onera.archives-ouvertes.fr/hal-01059010
Submitted on 29 Aug 2014

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L’archive ouverte pluridisciplinaire HAL, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d’enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.
Turbulence Model Comparison for Compact Plate Heat Exchanger Design Application

Francesco Vitillo¹,²,*, Lionel Cachon¹, Pierre Millan², Philippe Reulet², Emmanuel Laroche²
¹CEA Cadarache, ²ONERA Toulouse

Abstract

In the framework of the Gas-Power Conversion System for the Advanced Sodium Technological Reactor for Industrial Demonstration (ASTRID) project design, works done at CEA are focused on the design of the sodium-gas heat exchanger. Compact plate heat exchangers are indicated as the most suitable technology for such applications. An innovative compact heat exchanger geometry is proposed in this paper: its innovation consists in creating a 3D mixing flow. The proposed geometry has also very good mechanical resistance to high pressure gradients, being suitable for a large variety of flow applications. The flowfield inside such a channel is experimentally studied using the Laser Doppler Velocimetry (LDV) technique. The main velocity, the radial velocity as well as the Reynolds stresses are measured: data show the high level of flow mixing and the 3D flow pattern inside the channel. The experimental measurements are then used to validate turbulence models: in particular Reynolds-Averaged Navier-Stokes (RANS) equations are closed using both isotropic 2-equation isotropic eddy viscosity models and a Non-Linear Eddy Viscosity Model (NLEVM). Presented results represent the first step in the assessment of innovative high-performance compact plate heat exchangers that can be used to increase the plant efficiency as well as decrease the capital cost of the single component.

I. INTRODUCTION

The ASTRID project is a French project aiming to build a GEN IV sodium fast reactor to burn minor actinides, to extend the availability of natural uranium resources still maintaining a safety and security level at least equal to that of current generation nuclear reactors¹. The sodium-water chemical interaction being one of the major issues for a safe operation of a SFR, a gas Brayton cycle has been investigated² to de facto eliminate the occurrence of such an accident. In particular, the best suited gas was found to be the nitrogen, having the best trade-off between plant efficiency and technological use. A number of studies has been done³ to evaluate the R&D program necessary to develop the sodium-gas heat exchanger, which is the critical component substituting the steam generator.

Conceptual designs of compact sodium-gas heat exchanger (SGHE) have been proposed⁴ (Figure 1) based on existing technologies such as the PCHE and the PSHE. Nitrogen has been chosen as the reference gas for such an application⁵.
A preliminary investigation made at CEA (based on RANS CFD computations) showed that the proposed double channel compact heat exchanger provides a higher compactness (expressed as the thermal power per unit volume of the component) than a commercial PCHE for the same thermal power and pressure drop, as Figure 2 shows.

Hence, the need for a large database of fluid flow inside such a channel is of primary importance to validate the numerical model as well as the performance comparison with other heat transfer geometries.

The innovative channel is therefore the object of the present study. First of all some experimental measurements are shown that are a small part of the database collected. Then, based on the experimental data, a turbulence model validation is performed in order to determine the most appropriate turbulence model to reproduce the characteristics of the actual flow. Note that the goal of the present paper is not to describe and explain in detail the physics of the flow but rather to validate a model against acquired experimental data. Moreover, as a first step, the current work does not take into account heat transfer inside the innovative geometry, being a pure aerodynamic validation.

II. EXPERIMENTAL MEASUREMENTS

II.A. Experimental setup

In order to show the primary and secondary fluid motion as well as the boundary layer behavior both for the in-bend and the mixing-zone flow, a LDV has been evaluated as the best measurement technique. In particular, due to its capability to measure boundary layers, a 2-C LDV setup has been used to measure the principal and the radial velocity. The experimental setup has been assembled at the ONERA-Toulouse center. The measurement chain is shown in Figure 3:

The used technical means are a Spectra Physics Stabilité 2017 Laser, a TSI Colorburst Multicolor Beam Sepatator Model 9201 (both of them shown in Figure 4) to split blue and green beams used for the 2-C velocimetry and to provide the Bragg frequency shifting), an ISEL displacing system, a TSI Colorlink Plus Multicolor Receiver Model 9230 (to measure the laser reception frequency shifting) and a TSI FSA4000 Multibit Digital Processor (to provide digital data to the post-processing computer). The visualization particles are Di-Ethyl-Hexyl-Sebacat (DEHS) droplets.
Details on the laser beams are shown in Table I, whereas a view of the measurement volume is shown in Figure 5:

<table>
<thead>
<tr>
<th>Data/Color</th>
<th>Green</th>
<th>Blue</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wavelength [nm]</td>
<td>514.5</td>
<td>488.0</td>
</tr>
<tr>
<td>Beam Diameter [μm]</td>
<td>90.42</td>
<td>85.76</td>
</tr>
<tr>
<td>Beam intersection major dimension [mm]</td>
<td>1.32</td>
<td>1.25</td>
</tr>
<tr>
<td>Fringe Spacing [μm]</td>
<td>3.70</td>
<td>3.52</td>
</tr>
<tr>
<td>Bragg Frequency [MHz]</td>
<td>10</td>
<td>10</td>
</tr>
</tbody>
</table>

The test section is composed by structural aluminum parts with glass windows where the LDV measurement is performed (see Figure 5).

The working fluid is air at atmospheric temperature and pressure. The inlet velocity is 13 m/s, to have a correspondent inlet Reynolds number of 50 600.

II.B. Measurements description

The full measurement campaign has been done on several channel cross sections in the bends as well as in the mixing zone. For the sake of space we will only discuss about a few of them, specifically two profiles in a bend and one profile in the mixing zone. The in-bend profiles lie in a plane having a 45° angular rotation from the bend inlet section. Refer to dotted lines in Figure 6 for their definition of line #1 and line #2:

II.C. Uncertainty evaluation

Three types of uncertainties have been identified during the experimental campaign, specifically the uncertainty due to the data acquisition chain, the uncertainty due to environmental conditions and the uncertainty due to the measurement volume position. Regarding the data acquisition chain uncertainty, the global system calibration gave a velocity standard deviation value never greater than 0.25% of the measured velocity. In order to evaluate the experimental uncertainties in our measurements, two specific tests have been made: first, to be able to quantify the influence of external parameters such as low frequency velocity fluctuations, optical quality of the test section all over the measurements, in-day temperature variations etc. we performed a repeatability test consisting in repeating several times (i.e. around 400 times for the in bend flow and around 200 times for the mixing zone flow) the measurement at the same position. Once collected this data, the uncertainty in the velocity has been calculated as the standard deviation of the sample with regard to the test mean value. Finally, to evaluate the uncertainty due to the measurement volume position, measurements have been done in the vertices of a cube centered in the reference point having the vertices at ± 0.2 mm from the center along the three axes X, Y, Z.

The final velocity uncertainty is calculated adding the three obtained variance, calculated as the 3σ where σ is the combined standard deviation determined by adding the variances of each source of uncertainty.

The final uncertainty analysis is shown in Table II and Table III. Note that the evaluation is strictly valid only for the points where the repeatability and the position uncertainty tests have been performed; nevertheless it gives a good idea of the global uncertainties for the actual flow.
TABLE II
Velocity U Uncertainty Analysis

<table>
<thead>
<tr>
<th>System Chain</th>
<th>σ Repeatability</th>
<th>σ Position</th>
<th>Total 3σ Uncertainty</th>
</tr>
</thead>
<tbody>
<tr>
<td>In-bend</td>
<td>0.03 m/s</td>
<td>0.01 m/s</td>
<td>0.13 m/s</td>
</tr>
<tr>
<td>In-mixing zone</td>
<td>0.01 m/s</td>
<td>0.07 m/s</td>
<td>0.26 m/s</td>
</tr>
</tbody>
</table>

TABLE III
Velocity W Uncertainty Analysis

<table>
<thead>
<tr>
<th>System Chain</th>
<th>σ Repeatability</th>
<th>σ Position</th>
<th>Total 3σ Uncertainty</th>
</tr>
</thead>
<tbody>
<tr>
<td>In-bend</td>
<td>0.001 m/s</td>
<td>0.14 m/s</td>
<td>0.45 m/s</td>
</tr>
<tr>
<td>In-mixing zone</td>
<td>0.01 m/s</td>
<td>0.07 m/s</td>
<td>0.26 m/s</td>
</tr>
</tbody>
</table>

TABLE IV
uw Reynolds Stress Uncertainty Analysis

<table>
<thead>
<tr>
<th>System Chain</th>
<th>σ Repeatability</th>
<th>σ Position</th>
<th>Total 3σ Uncertainty</th>
</tr>
</thead>
<tbody>
<tr>
<td>In-bend</td>
<td>0.06 m²/s²</td>
<td>0.05 m²/s²</td>
<td>0.26 m²/s²</td>
</tr>
<tr>
<td>In-mixing zone</td>
<td>0.04 m²/s²</td>
<td>0.14 m²/s²</td>
<td>0.72 m²/s²</td>
</tr>
</tbody>
</table>

Note that the position uncertainties are quite high and are responsible for a large part of the total uncertainty: indeed these high values are due to the high velocity gradients (hence the large amount of turbulence) in the fluid flow.

III. NUMERICAL VALIDATION

III.A. Turbulence model description

Three turbulence models have been used to test their ability to correctly represent the fluid flow inside the channel. In particular two isotropic eddy viscosity models (e.g. Realizable k-ε and k-ω SST), and a NLEVM will be compared. FLUENT ® code has been used to perform calculations. Since the eventual goal of the study is to use a cost-effective numerical model for the design the innovative heat exchanger, only two-equation RANS model are evaluated. Hence more complex models such as the Reynolds Stress Transport (RSM) models are out of the scope of this work.

The Realizable k-ε model was proposed firstly by Shih et al.7. The transport equations of the turbulent variables for an incompressible flow are:

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_e}\right) \frac{\partial k}{\partial x_j}\right] + G_k - \rho \varepsilon \tag{1}
\]

\[
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right) \frac{\partial \varepsilon}{\partial x_j}\right] + \rho C_1 S \varepsilon - \rho \varepsilon C_2 - \frac{\varepsilon^2}{k + \sqrt{\varepsilon}} \tag{2}
\]

\[
\mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon} \tag{3}
\]

Where \( G_k \) is the turbulence kinetic energy production term. To be consistent with the Boussinesq hypothesis:

\[
G_k = -\rho \mu_t \dot{u}_i \frac{\partial u_i}{\partial x_i} = \mu_t S^2 \tag{4}
\]

with \( S \) being the modulus of the mean rate of strain tensor, defined as follows:

\[
S = \frac{1}{\sqrt{2}} S_{ij} S_{ij} \tag{5}
\]

The model is closed by the eddy viscosity formulation, where \( C_\mu \) is defined as:

\[
C_\mu = \frac{1}{A_0 + A_\varepsilon \frac{k}{\varepsilon} U^*} \tag{6}
\]

Where \( U^* = \sqrt{S_{ij} S_{ij} + \Omega_{ij} \Omega_{ij}} \tag{7} \),

\[
A_0 = 4.04, \ A_\varepsilon = \sqrt{6} \cos \phi, \ \phi = \frac{1}{3} \cos^{-1}\left(\sqrt{6} W\right), \ W = \frac{S_{ij} S_{ij} k_{ij}}{s^3} \ \text{and} \ s = \frac{1}{\sqrt{6}} S_{ij} S_{ij} \tag{8}
\]

The coefficient \( C_1 \) is expressed as:

\[
C_1 = \max \left[ 0.43, \eta + \frac{5}{\eta} \right], \ \eta = \frac{S}{\sqrt{6} W} \tag{9}
\]

The model constants are:

\[
C_2 = 1.9, \ \sigma_k = 1.0 \ \text{and} \ \sigma_\varepsilon = 1.2.
\]

To allow for integration up to the wall a Two-Layer Formulation is used: in particular, given the turbulent
Reynolds number $Re_y = \frac{\rho v y}{\mu}$, for a $Re_y > 200$ the complete $k$-$\varepsilon$ model is employed, i.e. both the transport equation for $k$ and for $\varepsilon$ are solved. For $Re_y \leq 200$ only the $k$-transport equation is solved, whereas the turbulence dissipation rate is calculated by a mixing length $\ell_{\varepsilon}$ evaluation as follows:

$$\varepsilon = \frac{k^{3/2}}{\ell_{\varepsilon}}$$

The eddy viscosity is damped from the fully turbulent to the buffer region value by the following expression:

$$\mu_{t,\text{buffer}} = \rho C_p \ell_{\mu} \sqrt{k}$$

The turbulence Prandtl numbers for $k$ and $\omega$ are expressed in the following way:

$$\sigma_k = \frac{1}{\sigma_{k,1} + (1 - F_1) \sigma_{k,2}}$$

$$\sigma_\omega = \frac{1}{\sigma_{\omega,1} + (1 - F_1) \sigma_{\omega,2}}$$

The function $F_1$ is computed as:

$$F_1 = \tanh(\phi_1^+),$$

where $\phi_1 = \min\left[\frac{\mu_k}{\sigma_k}, \frac{\mu_\varepsilon}{\sigma_\varepsilon}\right]$ and $D_\omega^+ = \max\left[\frac{1}{2} \frac{\partial k}{\partial \omega} \frac{\partial \omega}{\partial x_i} \frac{10}{10^{-10}}\right]$. The turbulence kinetic energy production term in the SST models has a limiter, defined as:

$$\overline{G}_k = \min(G_k, 10\beta^* k \omega)$$

The specific dissipation rate production is given by:

$$G_\omega = \frac{\mu}{\mu_\varepsilon} \overline{G}_k, \quad (23) \quad \alpha = F_1 \alpha_{\omega,1} + (1 - F_1) \alpha_{\omega,2}, \quad (24)$$

$$\alpha_{\omega,1} = \frac{\beta_{1,1}}{\beta^*} - \frac{\kappa^2}{\sigma_{\omega,1} \sqrt{\beta^*}}, \quad (25)$$

$$\alpha_{\omega,2} = \frac{\beta_{1,2}}{\beta^*} - \frac{\kappa^2}{\sigma_{\omega,2} \sqrt{\beta^*}} \quad (26)$$

The coefficient $\beta$ is expressed by:

$$\beta = F_1 \beta_{1,1} + (1 - F_1) \beta_{1,2} \quad (27)$$

The Cross-Diffusion term $D_\omega$ is expressed as follows:

$$D_\omega = 2(1 - F_1) \rho \frac{1}{\sigma_{\omega,2}} \frac{\partial k}{\partial \omega} \frac{\partial \omega}{\partial x_i} \frac{\partial x_j}{\partial x_j} \quad (28)$$

Finally the model is closed by the definition of the eddy viscosity:

$$\mu_t = \frac{\rho k}{\omega} \max\left[\frac{1}{\omega}, \frac{\gamma_\omega}{\gamma_k} \right] \quad (29)$$

$$F_2 = \tanh(\phi_2^+), \quad (30) \quad \phi_2 = \max\left(\frac{2\sqrt{k}}{0.09\omega y'}, \frac{500\mu}{\rho y^2} \right)$$

Model constants are:

$$C_{T,1} = 1.7, \quad C_{T,2} = 1.0, \quad C_{T,3} = 2.0, \quad C_{T,4} = 1.168, \quad \beta^* = 0.9, \quad \alpha = 0.3, \quad \beta_{1,1} = 0.075, \quad \beta_{1,2} = 0.0828$$

With regard to the NLEVM, the adopted model is that of Baglietto et al. (which has been implemented in FLUENT® via specific User Defined Functions). In particular this is a modified version of the Shih-Zhou-Lumley NLEVM10, where the $C_\mu$ value is no longer constant (like in the Realizable $k$-$\varepsilon$ model) but it is expressed as a function of the shear invariant due to realizability constraints:

$$C_\mu = \frac{2/3}{3.9 + S} \quad (32)$$

This time $S$ is the strain invariant, e.g.:

$$S = \frac{k}{\varepsilon} \frac{1}{2} S_{ij} S_{ij} \quad (33)$$
Moreover, the Reynolds stress tensor is expressed as follows:
\[
\overline{p'q'} = \frac{2}{3} \rho k \delta_{ij} - \mu_i \delta_{ij} + \frac{k}{\mu} \left[ \frac{1}{3} \delta_{ij} \Omega_{kl} + \frac{1}{3} \delta_{ij} \Omega_{kl} \right] + \frac{2}{\mu} \left[ \Omega_{ik} \Omega_{jk} + \Omega_{ik} \Omega_{jk} - \frac{1}{3} \delta_{ij} \Omega_{kl} \Omega_{kl} \right]
\]

Values of the average wall y+ are about 2.8, 1.4 and 0.8 respectively for configuration A, B and C. Meshing has been done using the ANSYS® Mesh module: an example of the inlet section mesh for the C configuration is shown in Figure 8:

![Figure 8 – Inlet section mesh for C configuration](image)

Three different meshing configurations have been preliminarily studied in order to evaluate the influence of mesh size on the solution. The three unstructured mesh configurations (hereafter named as A, B, C) differ by a factor of around 2 on the mesh size in the first layer thickness and in the global size of a single mesh cell:

<table>
<thead>
<tr>
<th>Configuration</th>
<th>First Layer Thickness</th>
<th>Major Size of a single Element</th>
<th>Number of Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>1x10^{-4} m</td>
<td>5x10^{-2} m</td>
<td>2,274,099</td>
</tr>
<tr>
<td>B</td>
<td>5x10^{-3} m</td>
<td>4x10^{-3} m</td>
<td>4,179,540</td>
</tr>
<tr>
<td>C</td>
<td>3x10^{-3} m</td>
<td>3x10^{-3} m</td>
<td>9,349,932</td>
</tr>
</tbody>
</table>

From values in Table VI see that configuration C shows a converged wall shear stress solution both for Realizable k-ε and SST model and a difference of about 1.5% in the average wall shear stress for the NLEVM model. Based on these trends, we retained configuration C as the reference meshing, a finer grid leading to numerically unstable calculations, especially for the NLEVM.
III.C. Comparison with experimental data

A first model validation is done on the channel pressure drop (Table VIII):

<table>
<thead>
<tr>
<th>LDV or Model</th>
<th>Value [Pa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>LDV</td>
<td>488 ± 20</td>
</tr>
<tr>
<td>k-ε Real.</td>
<td>557.4</td>
</tr>
<tr>
<td>SST</td>
<td>549.3</td>
</tr>
<tr>
<td>NLEVM</td>
<td>512.4</td>
</tr>
</tbody>
</table>

The NLEVM model seems to provide a good pressure drop evaluation. Nevertheless it is worth analyzing the collected LDV data in order to better understand the most appropriate model to simulate the flow inside the channel.

With regard to the geometrical details given in Figure 6 and Figure 7 we are going to show the measurements of the X-velocity (hereafter “U”), Z-velocity (hereafter “W”) and XZ-Reynolds stress (hereafter “uw”). Figure 9 to Figure 17 show the comparison between experimental LDV data and numerical calculations with the three models previously described. Note that abscissas are non-dimensional coordinates (Y* and Z*) for the region of interest. In particular for the mixing zone the middle-plane (i.e. the plane where mixing occurs) is in Y*=0.5.

Figure 9 – U velocity comparison in line #1

Figure 10 – W velocity comparison in line #1

Figure 11 – uw Reynolds Stress comparison in line #1

Figure 12 – U velocity comparison in line #2
As a general consideration, see that the SST model give the best overall results compared to the other models. As regards the in-bend flow, the SST model seems to provide very good results. Principal U velocity as well as radial W velocity profiles are very well calculated. On the other hand the uw Reynolds stress is generally underpredicted, still having a good global trend. The Realizable k-ε and the NLEVM model cannot reproduce the actual flow as precisely as the SST model.

The principal reasons are to be found in the fact that the $\varepsilon$-based formulation of the Realizable k-ε and the NLEV models is known not to work properly for adverse pressure gradient flows$^{11}$. Moreover the SST eddy viscosity formulation is known to give very good results for adverse pressure gradient boundary layers: in fact, even though the NLEVM model has an improved eddy viscosity formulation (i.e. through the variable $C_\mu$), it is supposed to assure global model realizability rather than improve adverse pressure gradient boundary layers. For an in-bend flow this seems to be a critical deficiency. See Figure 18 to analyse the eddy viscosity distribution in the outer bend boundary layer in line 1:
While the eddy viscosity in the viscous sublayer is quite the same for the three models (being a small fraction of the molecular viscosity in this region), differences appear in the outer boundary layer. There is a factor of about two between the eddy viscosity calculated by the Realizable k-ε and the NLEV models and the one calculated by the SST. See indeed the high velocity gradient eddy viscosity limiter in the SST formulation, which seems to act very well in the near wall region; this is not necessarily true for the Realizable k-ε and the NLEV models. Hence it appears that the formulation of the eddy viscosity used in a very similar way in the Realizable k-ε and the NLEV model (Eq. 6 and Eq.32) gives very similar results but does not work properly, at least for adverse pressure gradient flows.

As regards the mixing zone, first of all note that there are two isolated points in the boundary layer at Y*≈1 both in Figure 15 and Figure 17: they are likely spurious experimental points where the LDV system measured a false value either due to proximity to the glass window or to a DEHS spot deposed in the glass window. Anyway the SST model correctly represents the two velocity components U and W and gives a good trend for the uw Reynolds stress. Concerning the Realizable k-ε and the NLEV model, the velocity profiles are not in good agreement with the experimental data and the uw stress, although it is good regarding the global range of values, does not have the right trend in the near middle-plane (Y*=0.5) region as well as in the near-wall region. A possible explanation for this behaviour likely lays again in the turbulence kinetic energy production term (Eq.29): this production limiter is supposed to make the model quasi-realizable, setting a necessary upper bound on the turbulence kinetic energy production term as shown by Durbin. See in Figure 19 the turbulence kinetic energy production profiles in the three models:

Together with the turbulence kinetic energy production term for the three used models, a Realizability plot is also shown, based on the upper bound proposed by Durbin and Park and Park. An actual improvement in the modelling of this term seems to be the limiter in the SST model. This limiter is supposed to improve the model behaviour in stagnation points: indeed the mixing zone middle plane seems to represent a zone of turbulence kinetic energy build-up due to the counter-converging two flows. Indeed all the three models respect the Realizability condition; nevertheless, the SST model is the only one that do not reach the Realizability-imposed limit in the middle of the mixing zone. Therefore the higher turbulence kinetic energy production both for the Realizable k-ε and the NLEV model in this zone provides an excessive turbulence kinetic energy, which gives a too much high value of the eddy viscosity along the Y direction: this results in a higher flow diffusion along the X and Z directions limiting the flow penetration along Y direction. This finally results in a more typical “all-alone” flow profile (e.g. not enough mixing), as those of Figure 15 and Figure 16.

IV. CONCLUSIONS

From this analysis it can be stated that the SST model gives very satisfactory results for the complex flow presented, even being a RANS two-equation model: it provides a good description of the fluid flow and a slight pressure drop overestimation. The Realizable k-ε almost never gives the good trends, still suffering the well-known deficiencies of all k-ε based models. The NLEV formulation coupled with the Standard k-ε model works slightly better than the Realizable k-ε model, giving a good prediction of the pressure drop but still presenting many deficiencies regarding the fluid flow description. In particular the major features providing such good results for the SST model seem to be the good in-bend flow description given by an ω-based model (like the SST), the turbulence kinetic energy production limiter and the eddy viscosity formulation: they actually prevent the turbulence kinetic energy build-up that provides an erroneous prevision of both in-bend and mixing zone when
Realizable k-ε and the NLEV models are used. Moreover, despite the fact that an non-isotropic model would be supposed to give better results for such a complex flow, this seems not to be the case if the non-isotropic model cannot correctly represent the in-bend flow. Indeed its utilisation is strongly recommended for the description and the following numerical studies in the flow. Hence its utilisation is strongly recommended for the description and the following numerical studies in the present geometrical configuration.

ACKNOWLEDGMENTS

This work is done within a PhD collaboration between the CEA-Cadarache center and the ONERA – Toulouse Center. The authors want to thank Mr. Christian Pelissier, Mr.Francis Micheli and Mr.Jean-François Breil for the valuable help in setting and operating the LDV system.

NOMENCLATURE

CFD: Computational Fluid Dynamics
ASTRID: Advanced Sodium Technological Reactor for Industrial Demonstration
GENIV: Generation IV International Forum
SFR: Sodium-cooled Fast Reactor
PCHE: Printed Circuit Heat Exchanger
PSHE: Plate Stamped Heat Exchanger
2-C: 2 component (i.e. of velocity vector)
LDV: Laser Doppler Velocimetry
σ: Standard deviation for a statistical distribution
U_i: i-mean velocity component
u_i: i-fluctuating velocity component
ρ: Fluid density
k: Turbulence kinetic energy (TKE)
μ: Molecular dynamic viscosity
μ_e: Eddy viscosity
ε: Turbulence dissipation rate (TDR)
σ_k: TKE Prandtl number
σ_ω: TDR Prandtl number
ω: Specific dissipation rate (SDR)
σ_{Pr}: SDR Prandtl number
y: distance to the nearest wall
Y+: non-dimensional distance from the wall

REFERENCES

6. L.CACHON et al., PATENT n°HD14482