CHEMICAL ENGINEERING TRANSACTIONS

VOL. 39, 2014

A publication of
AIDIC
The Italian Association of Chemical Engineering
www.aidic.it/cet

Guest Editors: Petar Sabev Varbanov, Jiří Jaromír Klemeš, Peng Yen Liew, Jun Yow Yong

ISBN 978-88-95608-30-3; **ISSN** 2283-9216

DOI:10.3303/CET1439153

On the Applicability of RANS-Turbulence Models in CFD Simulations for the Description of Turbulent Free Jets During Biomass Combustion

Martin Miltner*, Christian Jordan, Michael Harasek

Institute of Chemical Engineering, Research division Thermal Process Engineering - Computational Fluid Dynamics, Vienna University of Technology, Getreidemarkt 9/166, A-1060 Vienna, Austria
martin.miltner@tuwien.ac.at

The purpose of the current work is to investigate the flow behaviour of both a straight and a slightly rotating turbulent free jet by means of CFD and RANS-turbulence modelling. The comparison of simulation results with data from literature as well as own experimental findings is used to assess the applicability of the numerous available turbulence models for the description of the analysed flow. For the experimental analysis a three dimensional Laser-Doppler-Anemometry system is used to measure gas velocity and turbulence intensity. The results of this work clearly indicate that the quality of the CFD simulation strongly depends on the proper choice of the applied approach for turbulence description.

1. Introduction

Turbulent free jets are widely spread throughout chemical engineering processes. Industrial applications comprise jet pumps, burners, mixers or ejectors (Decker et al., 2011). These applications take advantage of certain characteristics of jet flow like intensified mixing, enhanced heat and mass transfer, entrainment and momentum exchange. Especially in the field of combustion of gaseous, liquid or solid fuels the application of turbulent free jets is numerously documented. While Miller and Tillman (2008) mainly focus on solid fuels, Keating (2007) provides a comprehensive introduction to the most important aspects of applied combustion of various fuels. A recently developed concept for combustion of bales of herbaceous biomass also strongly depends on the amenities of straight and slightly rotating turbulent free jets (Miltner et al., 2007).

Computational Fluid Dynamics (CFD) has shown to be a powerful tool during the development and optimisation of chemical engineering processes and involved apparatuses. One of the first and most delicate problems occurring during CFD modelling is the proper description of turbulence. The direct simulation of stochastic turbulent effects is numerically and computationally demanding and impractical for industrial-scale applications. Therefore, turbulence has to be modelled, most commonly in the way of solving Reynolds-averaged Navier-Stokes equations together with a model for the generation and transport of a certain turbulence property. Many of these turbulence models are available today, most of them especially developed to describe one particular type of flow with great precision (Wilcox, 1994). It is the responsibility of the CFD engineer to choose the most suitable turbulence model for the individual flow problem.

Turbulent free jets have already been experimentally examined in great detail. Wilcox (1994) has published measurements of velocity profiles for the straight turbulent free jet from a round nozzle initially performed by Bradbury in the year 1965. Ashforth-Frost and Jambunathan (1996) have investigated the influence of nozzle geometry and confinement on the potential core region of turbulent free jets. Another examination of the three-dimensional flow in the near-field region of round free jets has been performed by Warda et al. (1999) providing significant insight into the jet structure at lower Reynolds-numbers. Most recently, Decker et al. (2011) performed a combined experimental and simulative assessment of flow and turbulence in a confined coaxial jet flow. An experimental analysis of the far-field region of turbulent free

Please cite this article as: Miltner M., Jordan C., Harasek M., 2014, On the applicability of RANS-turbulence models in CFD simulations for the description of turbulent free jets during biomass combustion, Chemical Engineering Transactions, 39, 913-918 DOI:10.3303/CET1439153

jets including an assessment of mixing has been published by Darisse and co-workers (Darisse et al., 2013). The flow field in the transition region of a round free jet and the influence of Reynolds-number has been analysed by Fellouah et al. (2009). Finally, an extensive review on the experimental and computational studies of round turbulent jets is given by Ball et al. (2012).

2. Materials and Methods

In order to assess the applicability of CFD together with various turbulence models for the description of turbulent free jets a simple test case consisting of a straight and a slightly swirling free jet has been analysed. This assessment is based on the comparison of simulation results with experimental data as well as findings published in literature. The underlying methodology is presented in the following.

2.1 Geometrical and operational setup

An isothermal free jet (35 °C) from a round pipe nozzle with an inner diameter of 104 mm has been investigated. The applied air volume flow of 200 m^3_{STP}/h resulted in a mean velocity of 7.38 m/s and a Reynolds-number of 45,727 for the pipe flow. The pipe with 1,000 mm length reaches out into the stagnant void space from a solid back wall at a height of 1,000 mm above the laboratory floor. Thus, the resulting jet is assumed to be only very slightly confined. The considered flow volume is depicted in Figure 1 together with the relevant dimensions. This figure also contains the analysed swirling body for the generation of a slightly rotating turbulent free jet. The baffle inclination of 20 ° results in a swirl number of 0.243 according to the definition of Günther (1984). The analysed swirling body also contains a deflector plate covering exactly 50 % of the nozzle outlet. The straight free jet is released from an empty pipe ending. The point referred as (0,0) in the following analysis corresponds to the centre of the nozzle outlet plane.

2.2 Experimental procedure

An experimental test facility with the described geometric and operational characteristics has been operated in an open-space laboratory environment. Pipe flow velocity has been measured by a measuring orifice device as well as by a hot-wire anemometer. The gas volume flow has been controlled by a fan and a frequency converter. Gas temperature has been assessed by a temperature probe (Pt100). The flow field and turbulence parameters have been measured with LDA equipment LASERVEC LDP100/IFA600 by TSI Corporation (Laser-Doppler-Anemometry). Spatial positioning of the measurement system and assessment of different velocity vector components have been conducted by traversing and rotating the LDA equipment. Glycerol fog has been used for seeding purposes.

2.3 Numerical modelling and simulation

CFD simulations have been performed applying the commercial solver FLUENT© 6.3.26 on a Linux-Cluster-hardware. The computational grid contains between 4.2 and 9.6x10^5 hexahedral volume cells with a strong structural refinement around the nozzle outlet. Solid walls have been modelled applying the non-slip wall condition and the flow properties in the wall-adjacent volume cell have been described with various wall functions (standard, non-equilibrium and enhanced wall functions). In order for this approach to be valid, the so-called y^+-criterion for the wall-adjacent cells has been maintained (Wilcox, 1994). Outlet boundaries have been modelled to feature a constant and uniform gauge pressure of 0 Pa. A uniform mass flow inlet with constant material properties of air at 35 °C is applied. Turbulence effects have been treated with the RANS-approach applying a number of well-known turbulence models of first and second order. The current implementation of the FLUENT© solver provides a number of sub-options for each of the turbulence models which also have been investigated. A list of all analysed turbulence models together with their individual computational demand relative to a laminar reference calculation is given in Table 1.

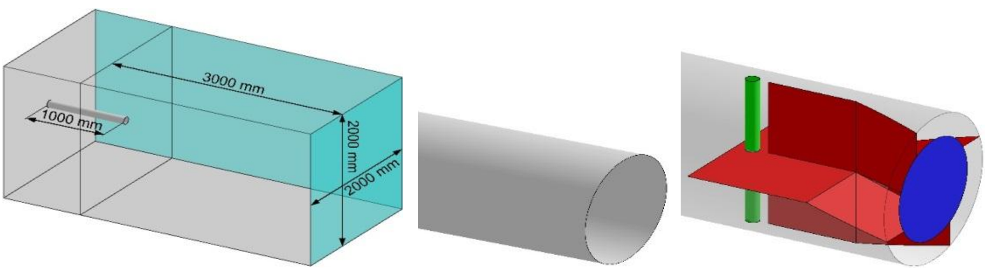

Figure 1: Geometrical overview of the analysed free jet test case (left), empty pipe exit for straight jet (centre), swirling body with 20 ° baffle inclination and deflector plate covering 50 % of the nozzle outlet (right)

Table 1: Analysed turbulence models together with relative iteration times

Turbulence model	Available sub-options	Relative iteration time
Reference case: laminar flow	-	1.00
Spalart-Allmaras	2	1.41
Standard k-ε	1	1.38
Renormalisation-Group k-ε	2	1.48
Realizable k-ε	1	1.56
Standard k-ω	2	1.31
Shear-Stress-Transport k-ω	1	1.58
Reynolds-Stress-Model	3	2.46

As no time-dependent effects were to be investigated only stationary CFD simulations have been performed. Solution convergence has been evaluated by monitoring scaled residuals and a number of flow parameters at relevant points of the computational domain.

2.4 Evaluation criteria

Assessment of applicability of simulation models for the description of the analysed test case is based on the three-dimensional gas flow field and the turbulence intensity field. Axial, radial and tangential (where applicable) gas velocity components have been investigated along the jet axis and across a number of vertical and horizontal profiles orthogonal to the jet axis (at different distances from the nozzle outlet). Only axial gas velocity field is presented in the current work, radial and tangential components are beyond the scope of this article. Turbulence intensity field has been chosen to be analysed in the same way as axial velocity components as this parameter was experimentally accessible. Only the most interesting results are presented in the current work in order to enhance conclusion clarity. This means, that only a given selection of all analysed turbulence models will be presented in the following.

3. Results

The results of the current work indicate that most turbulence models provide quite reasonable prediction of the flow field of the analysed turbulent free jets. Only the Standard k-ω-model obviously gives non-physical results. Nevertheless, pronounced differences in the quality of results delivered by the available turbulence models are clearly observable. Given the importance of turbulence effects for CFD simulations in chemical engineering practise, the selection of appropriate turbulence models is highly recommended. Figure 2 shows the dimensionless axial flow velocity along the dimensionless jet axis for both free jets. The near-field of the straight jet is determined by a region with almost constant centreline velocity (core region) followed by a transitional region and the jet far-field with a continuously dropping velocity proportional to 1/x (this region is also called principal region or self-preserving region). Most models provide good results in the near-field and transition region (exceptions Realizable k-ε and Standard-k-ω) but slightly underestimate gas velocities in the far field. Contrary to that, the results for the rotating jet seem to be somewhat more coincident. The near-field around the stagnant zone in the wake of the deflector plate and the jet far-field are reasonably well predicted, the gas velocity in the transient zone tends to be slightly overestimated by most models.
The far-field of the straight turbulent free jet (beginning at a distance between 10 and 14 nozzle diameters downstream the nozzle outlet) is characterised by a self-similar broadening of the jet (Wilcox, 1994). This means that profiles of axial velocity (related to the maximum velocity of the individual profile at the jet axis) plotted against the distance to the jet axis (related to the axial distance of the individual profile to the nozzle outlet) are coincident for arbitrary profiles within that region. This behaviour allows for a very condensed analysis of velocity profiles and provides an additional possibility to evaluate the physical strength of the simulation results of a certain turbulence model. Figure 3 shows these self-similar velocity profiles from experimental data collected at distances between 10 and 30 nozzle diameters (data from current work and data from Bradbury as published by Wilcox, 1994). Experimental data are accompanied by simulation results for a profile at a distance of 20 nozzle diameters as calculated by four different turbulence models. It can be seen that Spalart-Allmaras-results are poor whereas Standard- and Realizable-k-ε- as well as SST-k-ω-results are very promising. The results calculated by RNG-k-ε- and especially Standard-k-ω-models are also very poor. The performance of the Reynolds-Stress-Model is ambiguous. Profiles at low distances from the nozzle outlet show good agreement with experimental and literature data but more distant profiles (> 15 nozzle diameters) predict a more pronounced broadening of

the jet. The reason for this behaviour will be discussed later. An interesting fact is that all applied turbulence models, even the most suitable ones, predict a broader jet than experimentally observed.

Turbulence intensity can be calculated from the statistical distribution of experimental data provided by LDA measurements. Figure 4 compares experimental values with simulation results for both analysed turbulent free jets. The qualitative and quantitative agreement between measurement and simulation turns out to be remarkably good especially when considering the complex nature of turbulence and the strong simplification during turbulence modelling. The swirling jet shows a strong turbulence maximum near the nozzle outlet due to the disturbing effects of the deflector plate. Far downstream the nozzle the difference in turbulence behaviour between the two jets is levelling out. Also the straight free jet features a distinct turbulence intensity maximum at the end of the core region (around 6 nozzle diameters downstream the nozzle outlet) and the position of this maximum is well-predicted by most of the analysed turbulence models.

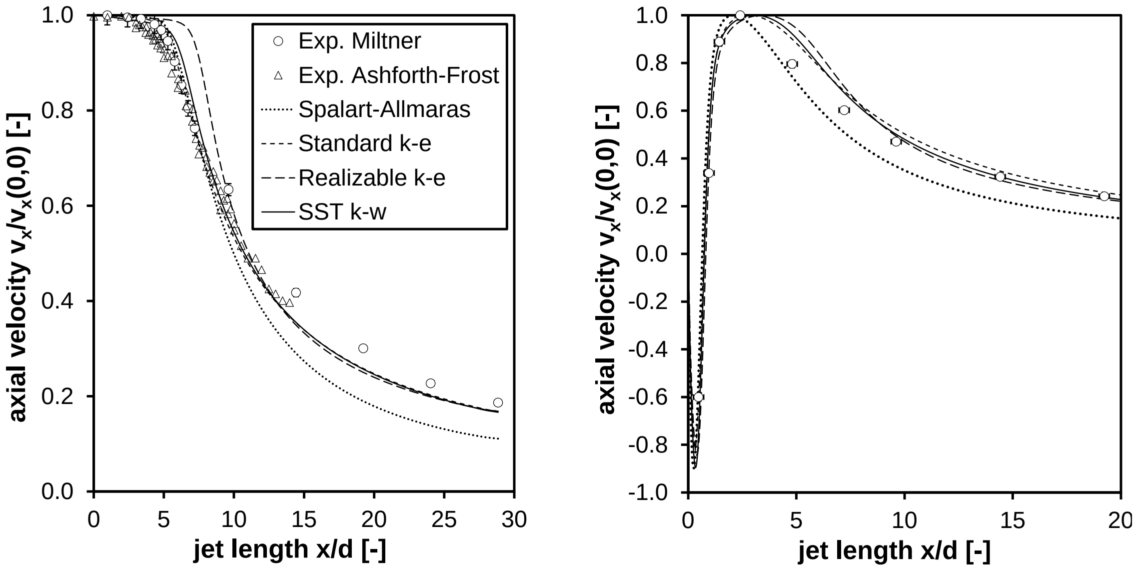

Figure 2: Dimensionless axial flow velocity vs. dimensionless jet length for straight jet (left) and slightly swirling jet (right): comparison of experimental data (Ashforth-Frost and Jambunathan, 1996) and CFD results achieved with different turbulence models

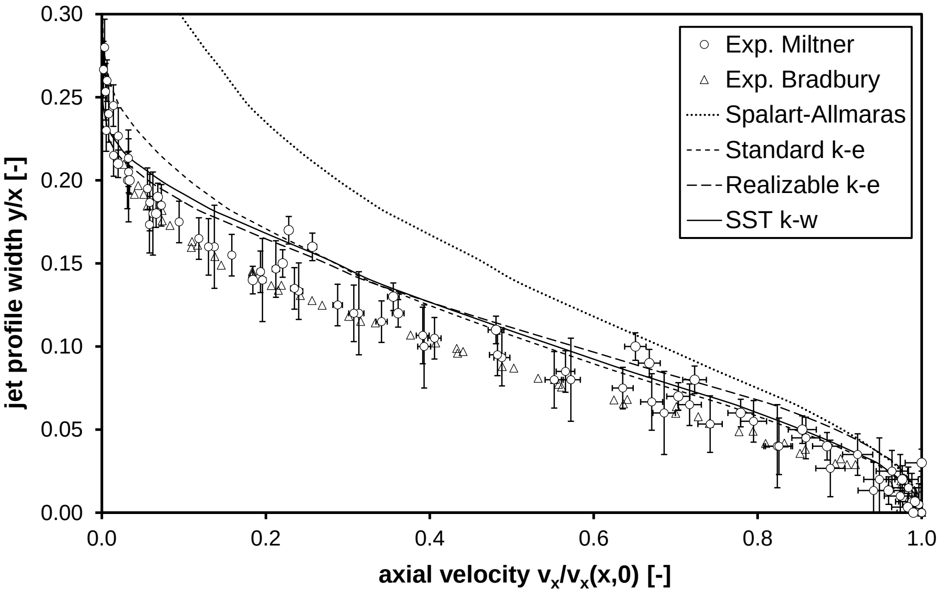

Figure 3: Dimensionless axial velocity profiles for the straight free jet: comparison of experimental data (Wilcox, 1994) and CFD results achieved with different turbulence models

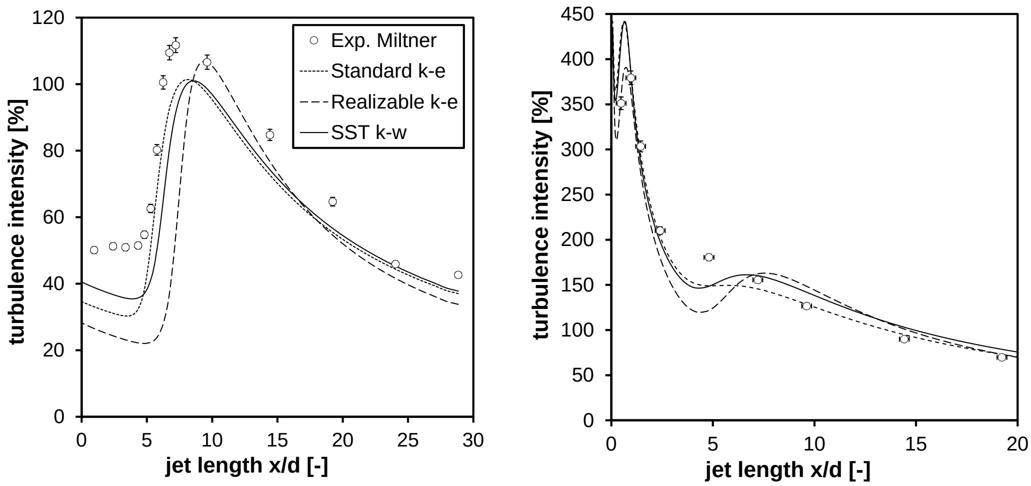

Figure 4: Turbulence intensity vs. dimensionless jet length for straight jet (left) and slightly swirling jet (right): comparison of experimental data and CFD results achieved with different turbulence models

Table 2: Comparison of experimental and simulation-derived values of characteristic jet parameters for straight and slightly swirling free jets from round nozzles

	Straight turbulent free jet		Lightly swirling turbulent jet
Turbulence model	Core length [-]	Spreading rate [-]	Spreading rate [-]
Experimental Miltner	5.21	0.095	0.118
Experimental (Wilcox, 1994)	-	0.086-0.095	-
Spalart-Allmaras	5.58	0.135	0.177
Standard k-ε	5.36	0.105	0.115
Renormalisation-Group k-ε	6.50	0.117	0.125
Realizable k-ε	7.10	0.105	0.115
Standard k-ω	4.53	0.131	0.125
Shear-Stress-Transport k-ω	5.75	0.107	0.116
Reynolds-Stress-Model	5.70	0.114	0.128

For a numerical comparison between simulation and experiments several characteristic jet parameters for both free jets have been calculated. Typically, the core length is defined by the point on the axis, where the flow velocity has dropped to an amount of 95 % of the initial velocity at the nozzle outlet. The termination of the core region occurs when the growing boundary layer between the jet and surroundings reaches the jet centreline (Tilton, 1997). Thus, the core length can also be defined to be the point of maximum turbulence intensity on the jet axis (Schlichting and Gersten, 1997). This termination is also obvious considering the maximum turbulence intensity level in the left diagram of Figure 4. The broadening of the jet is characterised by the so-called spreading rate. This parameter is derived by determining the single point at each flow profile where the axial velocity reaches the half of its profile peak value (Wilcox, 1994). The higher the number, the broader the evolving jet. The values of these parameters calculated from simulation results, own experimental findings as well as literature data are summarized in Table 2. The good agreement of own experimental results and values from literature (Wilcox, 1994) also complemented by Figure 3 proves reasonably high reliability of the experimental procedure presented in the current work. Data shows that all models predict a higher broadening of the jet than experimentally recorded. In this context, the results for the swirling jet are slightly more promising than the results for the straight jet. The most suitable models regarding this analysis are Standard-k-ε- and SST-k-ω-models. Thanks to its second order approach the Reynolds-Stress-Model is expected to be the most precise formulation (though computationally demanding) regarding turbulent flow. Unfortunately, the model is unable to unfold its full performance at this specific test case. While it is the most suitable model for the rotating jet we encountered only mediocre performance for the straight free jet. Especially with growing distance from the nozzle outlet the flow field predictions become notably worse. The reason for this observation is a strong dependency of the turbulence model on the boundary condition of a uniform pressure at the outlet faces. The distance between these homogenising boundary conditions and the flow regions of interest is too low for the Reynolds-Stress-Model. It is important to mention that the results of this model are not wrong. The uniform-pressure boundary condition does not exactly match the conditions encountered during the experiments. And if simulation results are influenced by this slight deviation the comparison is not strictly valid anymore. While the other turbulence models provide very similar results on a computational domain with doubled distances to the boundaries in all spatial directions, the Reynolds-Stress-Model provides significantly improved results for these cases. This behaviour has to be kept in mind when setting up similar CFD simulations.

4. Conclusions

It has been shown that CFD is able to serve as a capable tool for predicting turbulent jet flow provided a suitable model for turbulence description is applied. Nevertheless, significant errors can be introduced to any CFD simulation if inappropriate models are used. Turbulent free jets represent free shear flows for which the Standard-k-ε-model has been specifically developed. Therefore, this model is very suitable for the observed kind of flow. The increments to this model embodied by the RNG- and the Realizable-k-ε-models deteriorate the results to be expected. The Standard-k-ω-model has been specifically developed for wall-bounded shear flow resulting in a very bad performance for the free turbulent jet. To a certain extent this is also valid for the Spalart-Allmaras-model. The Shear-Stress-Transport-k-ω-model uses a formulation that applies k-ω-approach for near-wall-cells while k-ε-approach is used in the free flow region. This is the reason for the strong performance of this model even for a wider variety of flow classes. This performance is provided together with a very reasonable computational demand and is the reason for the widely-spread utilisation of the model. According to the physical nature of its model formulation, the Reynolds-Stress-Model is expected to be the best option for current RANS-turbulence simulations. This is beyond all question and has also been validated in recent studies (Decker et al., 2011). Nevertheless, the strong dependency of this model on the type and distance of boundary conditions for the computational domain resulted in a mediocre performance for the current analysis. It is important to mention that uniform pressure outlet boundary conditions can have a strong negative influence on simulations performed with this model.

References

Ashforth-Frost S., Jambunathan K., 1996, Effect of nozzle geometry and semi-confinement on the potential core of a turbulent axisymmetric free jet, International Communications in Heat and Mass Transfer, 23(2), 155-162.

Ball C., Fellouah H., Pollard A., 2012, The flow field in turbulent round free jets, Progress in Aerospace Sciences, 50, 1-26.

Darisse A., Lemay J., Benaissa A., 2012, Investigation of passive scalar mixing in a turbulent free jet using simultaneous LDV and cold wire measurements, Int Journal of Heat and Fluid Flow, 44, 284-292.

Decker R., Buss L., Wiggers V., Noriler D., Reinehr E., Meier H., Martignoni W., Mori M., 2011, Numerical Validation of a Coaxial and Confined Jet Flow, Chemical Engineering Transactions, 24, 1459-1464.

Fellouah H., Ball C., Pollard A., 2009, Reynolds number effects within the development region of a turbulent round free jet, International Journal of Heat and Mass Transfer, 52, 3943-3954.

Günther R., 1984, Combustion and furnaces; Springer Verlag, Berlin, Germany, (in German).

Keating E.L., 2007, Applied Combustion, 2nd Edition, CRC Press, Boca Raton, USA.

Miller B., Tillman D.A., 2008, Combustion Engineering Issues for Solid Fuels, Academic Press, San Diego, USA.

Miltner M., Miltner A., Harasek M., Friedl A., 2007, Process simulation and CFD calculations for the development of an innovative baled biomass-fired combustion chamber, Applied Thermal Engineering, 27, 1138-1143.

Schlichting H., Gersten K., 1997, Boundary-layer theory , 9th Ed, Springer Verlag, Berlin, Germany) in German.

Tilton J.N., 1997, Perry´s Chemical Engineers´ Handbook, 7th edition, section 6, page 6-20, McGraw-Hill, New York, USA.

Warda H.A., Kassab S.Z., Elshorbagy K.A., Elsaadawy E.A., 1999, An experimental investigation of the near-field region of free turbulent round central and annular jets, Flow Measurement and Instrumentation, 10(1), 1-14.

Wilcox D.C., 1994, Tubulence Modeling for CFD, 2nd Edition, DCW Industries, La Canada, USA.