Proceedings of the 4th WSEAS International Conference on Fluid Mechanics and Aerodynamics, Elounda, Greece, August 21-23, 2006 (pp148-153)

Prediction of Liquid Jet Pump Performance Using Computational Fluid Dynamics

M.D. EL HAYEK[+], A.H. HAMMOUD*
([+]) Mechanical Engineering Department, Notre Dame University, Zouk-Mosbeh, LEBANON
(*) Mechanical Engineering Department, Beirut Arab University, Beirut, LEBANON

Abstract: The present investigation deals with the application of advanced numerical techniques to the prediction of the overall performance (head ratio and efficiency) of liquid jet liquid pumps. The task is carried out by solving the flow equations, along with two turbulence models, namely, the k-ε model and the Reynolds stress model, using a finite volume approach with the appropriate boundary conditions. Comparison with experimental results is performed and proper conclusions are drawn as for the suitability of CFD techniques to the problem under investigation and their possible contribution to the improvement of the overall design. The classical analytical model is also included in the comparison and some recommendations concerning the different loss coefficients usually used in such approximate models are given.

Key-Words: Ejector, Eductor, Jet Pump, Computational Fluid Dynamics, Turbulence Models, Mixing.

1 Introduction

Jet pumps or eductors/ejectors are mechanical devices used to transfer energy from a primary stream of fluid to a secondary stream. The latter may be of any kind: liquid, gas, or a mixture of liquids, gases and solids.

Jet pumps are used in practice for their versatility or ability to accommodate any fluid as well as a set other features like simplicity, reliability (no moving parts), low cost, … Their main drawback is their relatively low efficiency due to the many losses experienced by the flow: mixing losses and friction losses in the primary nozzle, the throat, the diffuser, and the suction chamber.

Careful studies are required to investigate possible ways of reducing the losses and as a result improving the overall design of jet pumps. Among these, the experimental approach was and remains a must. However, the parameters, which affect the operation of jet pumps, are numerous and any experimental program aiming at investigating their effects has to involve a large number of variations in the design and even a larger number of trials. As a result, the experimental approach is very expensive and usually not affordable all the time. This is reflected by the few publications concerning the subject, especially when dealing with liquid jet pumps [1,2]. More recent experimental investigations have confirmed the poor overall performance of jet pumps [3,4].

A modern and much cheaper approach is the computational technique as based on advanced numerical methods and powerful computing facilities. Computational methods are nowadays so reliable that they are widely used even at the design stage. They are ideally suited for the problem under investigation due to their many advantages over the traditional experimental techniques. They are in general universal techniques and can be adapted very quickly to any change or improvement that may arise during the design process.

Jet pump flows are very complex and require special computational techniques able to cope with the various physical aspects encountered in such flows: mixing, jet flow, recirculation, pressure recovery, etc… The problem is even more complicated when the multiphase nature of the flow is considered since jet pumps are most often used to carry a combination of media (liquid/solid, …). This may explain the little interest in the problem among the specialists in CFD [5,6]. Furthermore, the CFD investigations are most often dedicated to the case of steam ejectors owing to their importance in steam plant design.

The present study is aiming at applying CFD technology to the prediction of the overall performance of liquid jet pumps, mainly the efficiency and the head ratio. It is restricted to the case of liquid jet liquid pumps with water as the driving and suction fluids. The main objective here is to assess the capabilities of standard CFD techniques in regards to the complex flow encountered in liquid jet pumps. To this end, a comparison with experimental results [2,3] is carried out and proper conclusions are drawn as for the reliability of the CFD approach, its appropriateness to the case being considered, and its ability to pinpoint problems very hard (and/or being very expensive) to detect using standard experimental methods.

2 Modeling Techniques

The typical liquid jet pump model sketched in fig.1 and used in [3] to carry out the experimental approach is adopted in the present investigation. It consists of a main cylindrical chamber or suction chamber, made of Plexiglas, of 10.1 cm diameter caped by two flanges through which proper openings are made in order for the fluid to flow in and out. A secondary port is used to carry the suction fluid to the main chamber where it is mixed with the main jet issuing from a nozzle of 7 mm diameter and both streams flow out through the collector or mixing area connected to a throat of 12 mm diameter used to continue the mixing process, and then a diffuser of 7° angle to recover the maximum amount of pressure possible.

Water is used as the working fluid for both the driving medium and the suction or driven medium. The driving flow rate is fixed at 47 l/min and the secondary or suction flow rate is varied from 0 to 47 l/min to mimic the experimental conditions. It is to be noticed that the problem being investigated numerically differs slightly from the experimental one by the fact that in the numerical approach, the driving and suction flow rates are assumed and both the driving and suction pressures are determined while in the experimental approach, the driving pressure and flow rate are fixed and the other quantities are measured. Having the flow rates, average velocities are calculated and used as velocity boundary conditions at the corresponding inlet.

2.1 Analytical Model

The overall jet pump characteristics can be evaluated using a set of simple equations representing the various conservation laws as applied to the different zones of the flow. Such an approach is extensively described in [1] and is based on three fundamental dimensionless groups, namely, the flow ratio or the ratio of the suction and driving flow rates ($M = Q_s / Q_j$), the head ratio or the ratio of the discharge/suction and driving/discharge pressure differentials ($N = (P_d - P_s)/(P_j - P_d)$), and the efficiency or the product of the flow ratio and the head ratio ($\eta = MN$).

The resulting model giving the head ratio in terms of the flow ratio is given by

$$N_n = 2b + \left(\frac{Mb}{1-b}\right)^2 \left[1 - 2b - K_{en}\right] - \left(1 + K_{td}\right) b^2 \left(1 + M\right)^2$$

$$N = \frac{N_n}{1 + K_n - N_n} \tag{1}$$

where b is an aspect ratio defined as the ratio of the nozzle cross-sectional area to the throat cross-sectional area. K_{td}, K_{en}, and K_n are loss coefficients for which the recommended values are 0.2, 0, and 0.05, respectively [1].

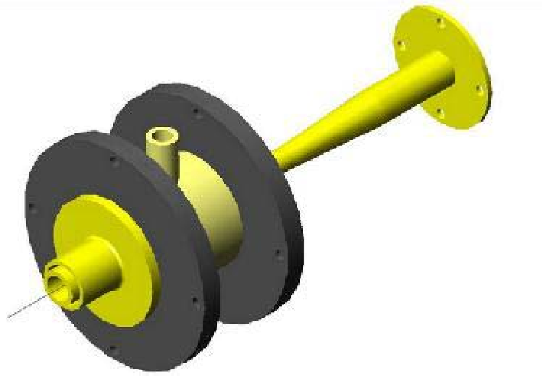

Fig.1, Jet pump model.

2.2 Governing Equations

The most appropriate equations to describe flow fields are the classical Navier-Stokes equations. For turbulent flows, as it is the case in the present application, however, and as a result of the complex nature of the flows being considered, a more practical representation is the Reynolds averaged Navier-Stokes (RANS) equations. The RANS equations involve usually time-averaged quantities and are typical transport equations representing the conservation of mass, and momentum along the different space directions as given by

$$\frac{\partial \rho U_j}{\partial x_j} = 0 \tag{2}$$

$$\frac{\partial \rho U_j U_i}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\mu \frac{\partial U_i}{\partial x_j} - \rho \overline{u_i u_j}\right] - \frac{\partial P}{\partial x_i} + \rho g_i \tag{3}$$

where U is the velocity, P the pressure, ρ the density, μ the absolute viscosity, and g the acceleration of gravity. The subscripts, i and j, represent the space coordinates (x,y,z).

The main problem with such transformed equations is the fact that they involve more unknowns than equations, a problem generally referred to as the turbulence closure problem. An appropriate solution is required in order to close the set under consideration. Usually turbulence models in more or less elaborate forms are used to determine the unknown statistical second-order moments, namely, the Reynolds stresses $\rho\overline{u_i u_j}$.

Many turbulence models have been developed so far and are widely used, with more or less success, to predict different kinds of engineering turbulent flows [7]. The present study adopts two kinds of models: the first-order standard k-ε model and the second-order Reynolds stress model [8].

2.3 k-ε Turbulence Model

The two-equation k-ε model is very popular due to its simplicity and robustness. It is based on the generalized Boussinesq hypothesis used to determine the Reynolds stresses.

$$-\rho\overline{u_i u_j} = \mu_t\left[\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right] - \frac{2}{3}\rho k\delta_{ij} \tag{4}$$

where μ_t is the eddy viscosity evaluated using dimensional analysis as

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{5}$$

The turbulent kinetic energy, k, and its dissipation rate, ε, are given by their transport equations modeled as

$$\frac{\partial \rho U_j k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \rho\varepsilon \tag{6}$$

$$\frac{\partial \rho U_j \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{\varepsilon}{k}\left(C_{\varepsilon 1}P_k - C_{\varepsilon 2}\rho\varepsilon\right) \tag{7}$$

where P_k is the production of turbulent kinetic energy by the mean flow field given by

$$P_k = -\rho\overline{u_i u_j}\frac{\partial U_i}{\partial x_j} = \mu_t\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j} \tag{8}$$

The different constants are given in the following table 1.

Table 1, Standard k-ε model constants.

C_μ	σ_k	σ_ε	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$
0.09	1.0	1.3	1.44	1.92

2.4 Reynolds Stress Turbulence Model

The first order models are isotropic models not able to capture the complexity of highly non-isotropic flows like encountered in jet pumps. A better approach is to use a second-order modeling technique and the most widely used model at this level is the Reynolds stress model or RSM. Many versions of the model were developed. The present study makes use of the standard version with the slight modifications described in [8].

In the RSM framework, the stresses are modeled directly using a transport equation for each.

$$\frac{\partial \rho U_k \overline{u_i u_j}}{\partial x_k} = D_{ij} + P_{ij} + \Phi_{ij} - \frac{2}{3}\rho\varepsilon\delta_{ij} \tag{9}$$

where P_{ij} represents the generation by mean fields given by

$$P_{ij} = -\rho\overline{u_i u_k}\frac{\partial U_j}{\partial x_k} - \rho\overline{u_j u_k}\frac{\partial U_i}{\partial x_k} \tag{10}$$

D_{ij} is the diffusion term generally modelled using the following generalized gradient diffusion hypothesis (GGDH)

$$D_{ij} = C_c\frac{\partial}{\partial x_k}\left(\frac{k}{\varepsilon}\rho\overline{u_k u_l}\frac{\partial \overline{u_i u_j}}{\partial x_l}\right) \tag{11}$$

The next term or the pressure-strain correlation Φ_{ij} is the most important due to its role in distributing the turbulent energy between the different stresses. This is what gives the second order closures their better physical basis and makes them superior to their first order counterparts. In the framework of the standard Reynolds stress model, Φ_{ij} is modelled as the superposition of two effects: the slow part or turbulence-turbulence interaction and the rapid part or the mean strain-turbulence interaction.

$$\Phi_{ij} = \Phi_{ij}^{(1)} + \Phi_{ij}^{(2)} + \Phi_{ijw}^{(1)} + \Phi_{ijw}^{(2)} \tag{12}$$

where $\Phi_{ij}^{(1)}$ and $\Phi_{ij}^{(2)}$ are modelled using the "return to isotropy" and the "isotropization of production" models, respectively.

$$\Phi_{ij}^{(1)} = -C_1\frac{\varepsilon}{k}\left(\rho\overline{u_i u_j} - \frac{2}{3}\rho k\delta_{ij}\right) \tag{13}$$

$$\Phi_{ij}^{(2)} = -C_2\left(P_{ij} - \frac{1}{3}P_{kk}\delta_{ij}\right) \tag{14}$$

and $\Phi_{ijw}^{(1)}$ and $\Phi_{ijw}^{(2)}$ are modelled using the standard wall reflection models.

$$\Phi_{ijw}^{(1)} = C_{1w}\rho\frac{\varepsilon}{k}\left(\overline{u_k u_l}n_k n_l\delta_{ij} - \frac{3}{2}\overline{u_i u_k}n_j n_k - \frac{3}{2}\overline{u_j u_k}n_i n_k\right)F_\alpha(x_n) \tag{15}$$

$$\Phi_{ijw}^{(2)} = C_{2w}\left(\Phi_{kl}^{(2)}n_k n_l\delta_{ij} - \frac{3}{2}\Phi_{ik}^{(2)}n_j n_k - \frac{3}{2}\Phi_{jk}^{(2)}n_i n_k\right)F_\alpha(x_n) \tag{16}$$

F_α is a function of the normal distance to the wall, x_n, used to damp the contribution of the wall terms in the pressure-strain correlation in the core of the flow. In the present study, an enhanced form is proposed and used instead of the traditional form.

$$F_\alpha(x_n) = \mathrm{MIN}\left(\frac{C_\mu^{-1/4}}{\kappa} \frac{\left(\overline{uv}\right)^2}{k^{1/2}\varepsilon x_n}, \frac{C_\mu^{3/4}}{\kappa} \frac{k^{3/2}}{\varepsilon x_n} \right) \qquad (17)$$

The last term in the turbulent stress equation (10) shows the dissipation mechanism as modelled using the hypothesis of isotropic small scales. The resulting dissipation rate is calculated using a transport-like equation similar to that used in the k-ε model (8) and given by

$$\frac{\partial \rho U_j \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left[C_\varepsilon \frac{k}{\varepsilon} \rho \overline{u_j u_k} \frac{\partial \varepsilon}{\partial x_k} \right] + \frac{\varepsilon}{k}\left(C_{\varepsilon 1} \frac{P_{kk}}{2} - C_{\varepsilon 2} \rho \varepsilon \right) \qquad (18)$$

where P_{kk} is twice the production rate given by the corresponding equation in the k-ε model (9).

The standard Reynolds stress model involves a set of constants, which is generally determined using simple flow situations. The commonly accepted set as used in the present work is given in Table 2.

Table 2, Constants for the RSM model.

C_c	C_1	C_2	C_{1w}	C_{2w}	C_ε	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$
0.22	1.8	0.6	0.5	0.3	0.18	1.44	1.92

2.5 Numerical Technique

A general purpose CFD code with various turbulence models is used in the present investigation. The well known finite volume method is considered with different algorithms to handle the pressure-velocity coupling and several differencing schemes to discretize the convective transport terms. In the present study, the SIMPLEC algorithm has been adopted along with a first-order upwind scheme [8]. Such schemes are accurate enough (as will be shown in figures 5 and 6) to be used safely in practical engineering applications.

The computational domain consists of the inside space of the jet pump. An unstructured grid of nearly 4.8×10^5 tetrahedral cells is used to map the entire domain as sketched in fig.2. Such a grid is fine enough to have a grid-independent solution according to tests conducted with different grid levels.

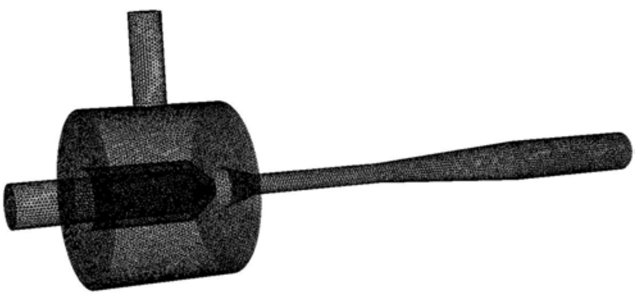

Fig.2, Meshed jet pump (483081 control volumes).

A standard wall function technique is used to handle the near-wall problem in both k-ε and RSM models. Velocities are specified at both inlets: the driving jet inlet and the suction flow inlet and outflow boundary conditions are considered at the exit.

3 Results & Discussion

A full set of simulations was carried out to evaluate numerically the different characteristics of the experimental jet pump. A first result is given in figures 3 and 4 where the velocity and pressure distributions are shown in a cut plane running along the axis of the pump. Here it is clearly noticed that the velocity is relatively large at the exit of the nozzle and as a result the pressure is very low. Actually the nozzle is the main responsible of the huge pressure drop in the pump from the inlet driving pressure to the exit or discharge atmospheric pressure. As expected, the suction pressure is even less than the discharge pressure but larger than the pressure prevailing in the suction chamber; a difference used to drive the secondary fluid into the chamber itself.

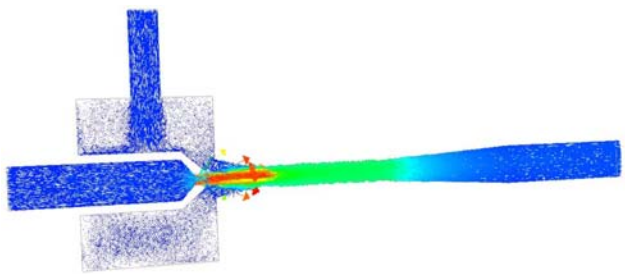

Fig.3, Velocity distribution in the vertical mid-plane (k-ε model, first order UDS, and flow ratio M = 0.7).

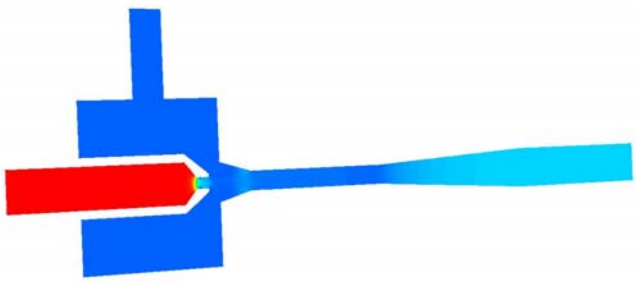

Fig.4, Pressure distribution in the vertical mid-plane (k-ε model, first order UDS, and flow ratio M = 0.7).

It is to be noticed also that the secondary jet is impinging directly on the pipe carrying the water to the nozzle. Such an impingement mechanism further contributes to the losses experienced by the pump under consideration. This may explain, along with other flow characteristics, the low efficiencies predicted both experimentally and numerically.

A quantitative representation of the same properties is shown in figures 5 and 6 where the centerline velocity and pressure are plotted along the axis of the pump. Here the sharp increase of centerline velocity and decrease of pressure are clearly shown. The very low pressure that appears in the suction chamber may become a problem in real life when cavitation is to be taken into account.

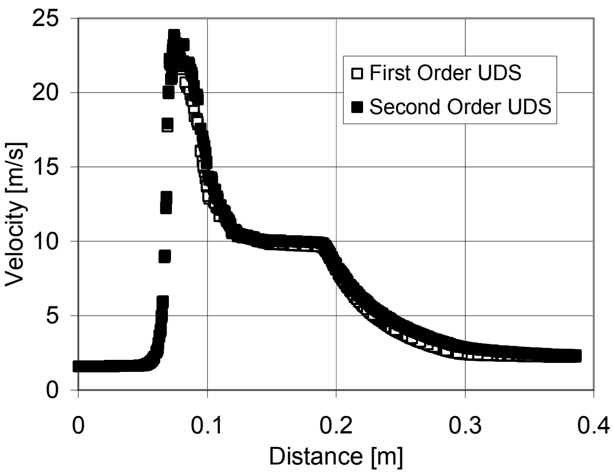

Fig.5, Velocity distribution along the axis of the pump as predicted by the k-ε model using first- and second-order upwind schemes.

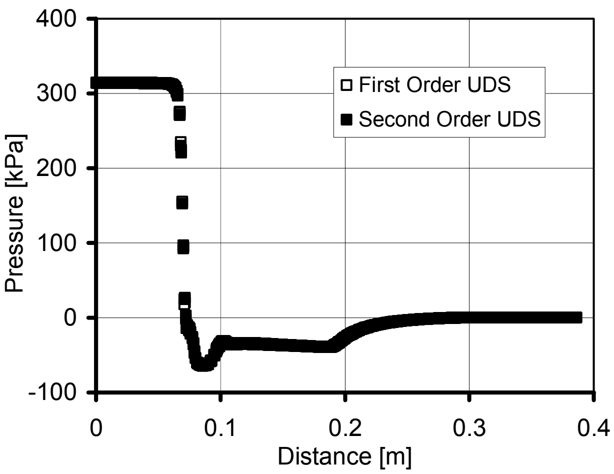

Fig.6, Pressure distribution along the axis of the pump as predicted by the k-ε model using first- and second-order upwind schemes.

The same figures show also a comparison between the results obtained with the first and second order upwind schemes. They are almost identical, which means that second–order schemes are not always the panacea, especially when dealing with practical engineering flow fields in which a lot of more serious problems are to be tackled.

Figures 7 and 8 show a comparison between the numerical results and the corresponding experimental and analytical solutions. It is clear, first of all, that the RSM turbulence model is better than the standard k-ε model. Such a result is expected since the k-ε model assumes the flow to be isotropic, an assumption far from being valid in the jet pump flow.

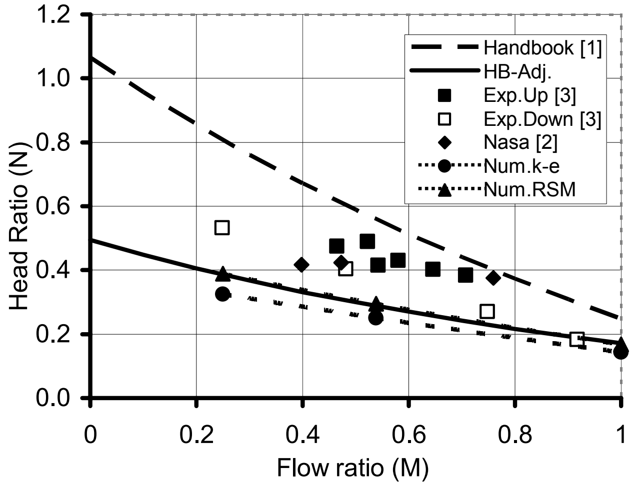

Fig.7, Pump head ratio calculated using experimental, analytical, and numerical techniques.

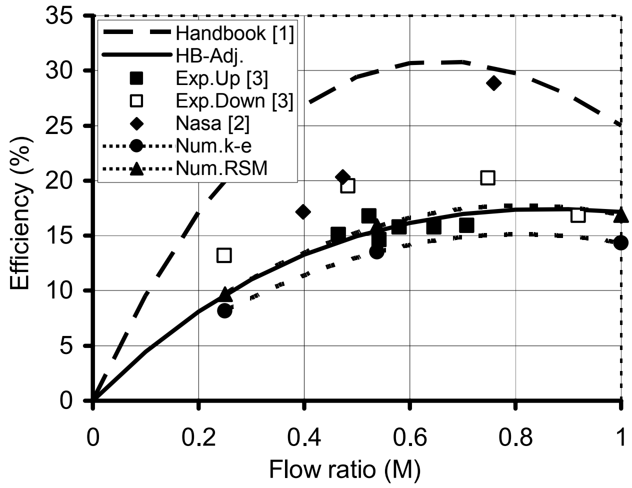

Fig.8, Pump efficiency calculated using experimental, analytical, and numerical techniques.

As for the experimental results, two sets are considered: Exp.Up corresponding to a positive suction mechanism in which the suction fluid is flowing downward and Exp.Down or negative suction mechanism (suction fluid flowing upward).

The negative suction is achieved in practice by rotating the pump by 180° around its axis. Few experimental points (Nasa) from Sanger [2] are also shown for validation purposes. Here, the agreement between numerical and experimental approaches is acceptable especially for the overall efficiency. It is important to notice at this level that the present numerical simulations are carried out with no contribution of gravity, e.g. no hydrostatic head either positive or negative to mimic truly the experimental set-up (positive/negative suction mechanisms). Such an approximation is generally acceptable due to the small size of the pump being considered and is generally not enough to explain the non-negligible difference in the experimental results between positive and negative suction mechanisms.

The analytical model with its standard set of loss coefficients as given in the pump handbook [1] seems to over-predict the behavior of the actual jet pump. Such an over-prediction may be explained by the simplified design of the experimental pump, which amplifies the losses as mentioned previously. A better agreement may be obtained (HB-Adj) if the loss coefficients are adjusted to 0.45, 0.55, and -0.5 for K_{td}, K_n, and K_{en}, respectively. The first two values are in line with the experimental range of variation given in the handbook ($0.17<K_{td}<0.4$ and $0.04<K_n<1.0$) while for the third, representing the losses in the throat entry, the negative value means simply that an improved way is needed in order to deal with the jet losses in the analytical model since negative loss coefficients are unphysical. Such an improved approach requires, however, additional data, either experimental or numerical. The ability of the adjusted analytical model to stick exactly to the numerical results is to be noticed. As it is shown both the slope and the magnitude are in perfect agreement with the RSM results

4 Conclusions

CFD techniques are common practice nowadays and the present study shows that they can be used in the field of jet pumps to provide insights concerning the physics of the flow and as a result to contribute to the possible improvement of the overall design. Detailed numerical tests are underway to investigate the effects of such or such departure from the standard design. The standard turbulence models need to be improved in order to reach a closer agreement between experimental and numerical results. Such an improvement is noticeable in the RSM results but still needs to be conducted further. More experimental data are also required since the existing data are characterized by a large scattering due to the many specific problems encountered during the implementation of the experiments.

References:

[1] R.G. Cunningham, Jet Pumps, in *Pump Handbook*, I.J. Karassik *et al.* (ed.), 3rd ed., McGraw-Hill, 2001.

[2] N.L. Sanger, Noncavitating Performance of Two-Low-Area Ratio Water Jet Pumps Having Throat Lengths of 7.25 Diameters, NASA TN D-4445, 1968.

[3] A.H Hammoud, Effect of Design and Operational Parameters on Jet Pump Performance, Personal Communication, 2006.

[4] Y. Yamazaki, A. Yamazaki, T. Narabayashi, J. Suzuki, T. Shakouchi, A Study on Performance of Jet Pumps: effects of Flow Passage Configurations, 6th World Conference on Experimental Heat Transfer, Fluid Mechanics and Thermodynamics, Matsushima (Japan), Apr. 17-21, 2005, Paper 4-b-9.

[5] I.W. Eames, A.E. Ablwaifa, Use of CFD in the Prediction of Jet Pump Performance, 3rd International Conference on Heat Powered Cycles, Larnaca (Cyprus), Oct.11-13, 2004.

[6] O.B. Kwon, M.K. Kim, H.C. Kwon, D.S. Bae, Two-Dimensional Numerical Simulations of the Performance of an Annular Jet Pump, *Journal of Visualization*, Vol.5, No.1, 2002, pp.21-28.

[7] D.C. Wilcox, *Turbulence Modeling for CFD*, DCW Industries, La Canada, California, 1993.

[8] M. El Hayek, *Le transfert de chaleur par convection en régime turbulent: aspects physiques et numériques*, Thèse de Doctorat Européen, Faculté Polytechnique de Mons, Mons, Belgique, 1997.