Investigation of Effects of Scale and Surface Roughness on Efficiency of Water Jet Pumps Using CFD

K. Aldaş & R. Yapıcı

To cite this article: K. Aldaş & R. Yapıcı (2014) Investigation of Effects of Scale and Surface Roughness on Efficiency of Water Jet Pumps Using CFD, Engineering Applications of Computational Fluid Mechanics, 8:1, 14-25, DOI: 10.1080/19942060.2014.11015494

To link to this article: https://doi.org/10.1080/19942060.2014.11015494
INVESTIGATION OF EFFECTS OF SCALE AND SURFACE ROUGHNESS ON EFFICIENCY OF WATER JET PUMPS USING CFD

K. Aldaş* and R. Yapıcı **

*Department of Mechanical Engineering, Faculty of Engineering, Aksaray University, 68100 Aksaray, Turkey
**Department of Mechanical Engineering, Faculty of Engineering, Selçuk University, Alaeddin Campus, 42070 Konya, Turkey.
E-Mail address: rafet@selcuk.edu.tr (Corresponding Author)

ABSTRACT:
Comparing to pumps with moving parts water-jet pumps have a lower efficiency and surface roughness is an important factor for these types of pumps. The aim of this simulation study is to numerically determine how the scaling-up, downscaling and change in the absolute and relative roughness would impact on the energy efficiency of the pumps, using a commercial computational fluid dynamics (CFD) solver ANSYS FLUENT. In order to select the turbulence model that produces the predictions closest to the actual data from four turbulence models, a preliminary study was conducted on a full-scale jet pump. Using the transition SST model, which gives the best results among all the models, the effects of scale and roughness on the performance of the pumps were investigated in the scale range from 1/4 to 20/1. The optimum efficiencies for different area ratios over a wide range were determined according to the scale and size of roughness. It was seen that the efficiency increases significantly up to a given scale size at a constant absolute roughness, while it is generally independent of the scale size at constant relative roughness. The relative efficiency for the area ratio 5.92 reduces to 60% at the relative roughness value of 0.05. Moreover, CFD appears to be the most appropriate tool for model studies of jet pumps.

Keywords: jet pump, scale effect, CFD, flow simulation, roughness effect

1. INTRODUCTION

Jet pumps are extraordinary compact devices that transport fluid by fluid. One of their most important features is that they provide the use of a centrifugal pump like another pump with a lower head, but with a higher capacity, thus resulting in a two or three fold increase in the flow rate. Fig. 1 shows a typical jet pump consisting of four main parts; the motive nozzle providing the strong vacuum, suction chamber into which is entrained the secondary (suction) flow, the mixing pipe in which the motive and suction streams are mixed with each other through the momentum exchange thus transferring to the sucked fluid energy from the motive fluid and diffuser which decelerates the mixing flow for pumping with a lower energy loss. In the high-efficiency types of these pumps, there is also a straightener providing the entry of symmetrically sucked flow into the suction chamber and nozzle.

Water jet pumps are widely used in various industrial applications, such as well-pumping, priming of centrifugal pumps, nuclear cooling, desalination systems, discharge of water from a valley, transport of solid material by pipelines, hydraulic dredging, slurry pumping, fuel pumping in airplanes and space shuttles and drip-irrigation. Efficiencies of these pumps are less than those of pumps with moving elements. Nevertheless, because of the absence moving parts, the pumps cost less and are more reliable. The probability of breakdown is lower and therefore, they require fewer repairs and less maintenance. In particular, water jet pumps are very suitable for handling dirty, corrosive and erosive fluids. However, these pumps rely on a pressurized liquid which is the equivalent of a rigid working part.

Through the turbulence models developed in recent years, the CFD simulation extends its application field and gives more reliable results. CFD is now widely used in the simulation of hydraulic turbomachines with complex geometry, such as a Francis turbine and a multistage centrifugal pump (Qian et al., 2011; Salvadori et al., 2012). The modern technique now provides numerical results with sufficient accuracy even in the optimization of gas jet pumps where strong shocks occur and the flow is supersonic (Li and Li, 2011; Hemidi et al., 2009; Fan et al., 2011).
For this reason, this technique can also be used for the design, analysis and testing of liquid jet pumps.

In the literature there are a large number of studies on analyzing jet pumps using CFD. Today, an important part of the studies on water jet pumps concerns the improvement of their efficiency and an enlargement of their application fields using flow simulation techniques together with experimental methods. The most remarkable of these studies is that of Narabayashi et al. (2006) who carried out a series of experimental and numerical simulation studies to improve the efficiency of water jet pumps used to re-circulate the reactor core coolant within the reactor high-pressure vessel in Boiling Water Reactor (BWR) plants. The researchers used a 1/5 scale model of the jet pump in the reactor in their experimental works and they preferred k-ε turbulence model in StarCD CFD code for their simulation studies. They investigated the effects of the motive nozzle number, the nozzle position, scale, surface roughness and the geometry of the mixing tube on the efficiency of the water jet pump. Based on their results, the best efficiency is obtained when the outlet of the motive nozzle is located in the inlet of the mixing tube and the efficiency decreases considerably with increasing roughness of the surface. Consequently, the authors indicated that by using a CFD code and model test, the efficiency of the jet pump would be improved.

Sun et al. (2011) used k-ε turbulence model available in FLUENT 6.2 in order to numerically study the effects of the convergence angle of the suction nozzle on the performance of a jet pump and the pressure distribution within the pump. They stated that if the angle was too small, cavitation could occur and thus, the pump efficiency would drop.

Long et al. (2008) numerically investigated the effects of the nozzle exit tip thickness on the performance and the flow field of a jet pump using k-ε turbulence model. According to their results, the thickness has only a little effect on the pressure ratio and efficiency of the pump, whereas
its effect on the flow field is relatively great.
In the study by Winoto et al. (2000), the pressure ratio and efficiency of water jet pumps were analyzed theoretically and experimentally as a function of the velocity ratio and the results obtained were compared. The authors tested a jet pump using three different driving nozzles (circular, square and triangular) in the pump with a constant area-mixing chamber. The best efficiency in their work was obtained with a circular nozzle.
Hayek and Hammoud (2006), predicted the performance of water jet pumps using the standard k-ε and RSM turbulence models. They compared their numerical results with the experimental and analytical results from the literature and stated that the RSM model gave better results than the k-ε model. The numerical results show general agreement with the experimental results, but the analytical results agree with experimental results only if the loss coefficients of the jet pump are adjusted to particular values. El-Sawaf et al. (2011) designed a special jet pump for pumping floating oils and constructed an experimental test rig to investigate the effects of various parameters on the performance of the pump. They obtained a lower efficiency than that of water jet pumps using oil in their experiments.
Yamazaki et al. (2006) investigated the effects of surface roughness of the mixing tube and the roughness location on the efficiency of a water jet pump. Their experimental results showed that roughness in the upstream region of the mixing tube has a greater effect on the efficiency than roughness in other locations. Moreover, the researchers determined that the mass flow ratio and the optimum efficiency ratio decrease linearly as the relative roughness increases.
In their research, Yamazaki et al. (2007) focused on the mixing of two flows with the aim of improving the efficiency of jet pumps. They attempted to determine their optimum type using jet pumps with different geometries. As a result of this work, they showed that the peak efficiency can be achieved using a jet pump with a single circular nozzle and a diverging pipe part in the downstream region of the mixing pipe.
Prakeao et al. (2002) analyzed the flow pattern and pressure distribution within a water jet pump using a three-dimensional RNG k-ε turbulence model. This research is focused on determining the optimum mixing tube length. Consequently, they stated that the maximum jet pump efficiency was obtained at an area ratio of 2.77 and a relative nozzle position of 0.5 when the relative mixing tube length was 3.5.
Regardless of the pump type, CFD simulation technique seems to be very attractive and affordable for modeling studies, and determining the performance of similar pumps of various sizes. The efficiency of the jet pumps, whether the pumped fluid is a liquid or a gas, is lower than any other commonly used rotodynamic and positive-displacement pumps or compressors. Therefore, their efficiencies should be improved or at least the pumps need to be operated in optimum conditions or at the highest possible efficiency within their existing configuration.
Water jet pumps generally operate under severe conditions, therefore, their inner surface roughness could be altered by various factors such as erosion, corrosion and incrustation. Even though these pumps are manufactured to a high standard, their surface roughness can reach very high values over time. On the other hand, since large-scale pumps may be more efficient, only one of such large-scale pumps operating alone in the same pumping system could offer more advantages than a group of small-scale pumps operating in parallel. The authors have not encountered any study on the efficiency of jet pumps over a large range of relative roughness, area ratio and scale.
The aim of this study is to determine whether the efficiency of a water jet pump can be improved by decreasing the relative roughness through scaling up or by reducing the absolute roughness size at a constant scale and, if possible, to perform an evaluation of the performance increment ratios of the improvement methods.
This purpose can be easily achieved using CFD, by contrast such a work is more expensive, and takes time and effort when using an experimental method. In the first part of the study in which a commercial CFD solver, ANSYS Fluent 14.0, is used, the performance and flow field of a full-scale jet pump with an area ratio of 5.92 and a nozzle outlet diameter of 15 mm are predicted using four RANS turbulence models. A comparison between the obtained numerical results from all the turbulence models and the experimental data from the literature (Yapici, 1990) shows that predictions of the transition SST model are slightly more accurate than those of other models.
In the second main part of the study, the effects of scale and roughness on the efficiency are investigated using the transition SST turbulence model for scale models in the range of 1/4 -20/1 for the same pump. Also, the variation of the efficiency with the scale size in an area ratio range
of about 2-16.5 is obtained.

2. MODELING OF FLOW IN JET PUMP

2.1 Physical model

A schematic drawing of the 1/1-scale jet pump with a cylindrical mixing pipe which is the basis of the simulation study is shown in Fig. 2 and the main dimensions are given below the figure. The flow of all fluids through the inlet and outlet of the pump are in the axial direction. In this type of water jet pump, the entrained rotational fluid enters symmetrically to the suction chamber and then the suction nozzle. This suction type is the most important factor of efficiency improvement. The inner surfaces of the pump body can be assumed to be hydraulically smooth since the mixing pipe is made from a new drawn pipe (Kt=0.00015 mm) and the inner surfaces of the diffuser, motive and suction nozzles are machined much more precisely than the pipe. The 3-D solid modeling of the jet pump casings having various scales used in the CFD simulation were designed using Design Modeler software in an ANSYS Workbench platform.

2.2 Numerical model

The computations were performed with an academic version of the Fluent R14.0 CFD code (2009), which employs a finite volume discretization. The numerical simulation is undertaken on the basis of three-dimensional steady flow of water through the pump. The benchmark tests for the jet pump were conducted in order to determine the appropriate turbulence model to be used in the CFD analysis. In this comparison study, the realizable k-ε, RSM, SST k-ε and transition SST models were used. After the tests all the simulation results were compared with the experimental data and it was concluded that the transition SST model provided more accurate results than other models that were tested. In all the computations, continuity and momentum equations were used together with the model (transport) equations. The Reynolds-averaged continuity and momentum equations for incompressible flow are as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} = 0$$  (1)

$$\frac{\partial (\rho u_j)}{\partial t} + \frac{\partial (\rho u_j u_k)}{\partial x_k} = - \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_j}{\partial x_j} \right) + \tau_{ij}$$  (2)

where the Reynolds stress is:

$$\tau_{ij} = -\rho u_i u_j$$  (3)

The transition SST turbulence model is a four-equation turbulence model which is based on the coupling of the SST k-ω model with the γ-Reθ model. The four transport equations of the transition SST model are defined as follows. The transport equation for the intermittency γ, which is a measure of the probability that a given point is located inside the turbulent region, is:

$$\frac{\partial (\rho \gamma)}{\partial t} + \frac{\partial (\rho \gamma u_j)}{\partial x_j} = - \frac{\partial}{\partial x_j} \left( \mu \frac{\partial \gamma}{\partial x_j} \right) + \beta$$  (4)

where P1 and E1 are transition sources and P2 and E2 are destruction / relaminarization sources. The transport equation for transition momentum thickness Reynolds number $\tilde{Re}_{th}$, which indicates the transition onset criteria, is:

$$\frac{\partial (\rho \tilde{Re}_{th})}{\partial t} + \frac{\partial (\rho \tilde{Re}_{th} u_j)}{\partial x_j} = P_n + \frac{\partial}{\partial x_j} \left[ \sigma_n (\mu + \mu_*) \frac{\partial \tilde{Re}_{th}}{\partial x_j} \right]$$  (5)

where $P_n$ is the source term.

The transport equations for the turbulence kinetic energy k and the specific dissipation rate $\omega$ are:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \mu \frac{\partial k}{\partial x_j} \right) \frac{\partial u_j}{\partial x_j} - 
\frac{\gamma_{off}}{\nu_c} \left( \tilde{D}_k - \bar{D}_k \right) + \frac{\partial}{\partial x_j} \left[ \left( \mu + \sigma_{\omega} \mu_{eff} \right) \frac{\partial \omega}{\partial x_j} \right]$$  (6)

where $\tilde{D}_k$ and $\bar{D}_k$ are the production and destruction terms for the turbulence model and the terms are,

$$\tilde{D}_k = \min(P_k, 10 \rho \beta \ast k_{o})$$

$$\bar{D}_k = \min(\max(\gamma_{off} \cdot 0.1), 1.0) D_k$$

$$D_k = \rho \beta \ast k_{o}$$

(7a-c)

were $D_w$ and $C_{dw}$ are the dissipation term and the cross-diffusion term for $\omega$, respectively.

The relationships among the Reynolds stress, turbulent viscosity and turbulence equations in the transition SST model could be summarized as follows. Reynolds stresses for incompressible flow based on Boussinesq approximation and turbulent viscosity are defined as:

$$\tau_{ij} = -\rho u_i u_j$$

$$\mu = \frac{k}{\omega} \frac{1}{\max \left( \frac{1}{a_1}, \frac{1}{a_2 \omega} \right)}$$

(9a-b)

where $S, F_2, a_1$ and $a_2$ are the strain rate magnitude, a blending function, the model constant and the turbulent viscosity damping factor, respectively.

The production of turbulent kinetic energy in the
standard k-ω model is given by
\[ P_1 = -\rho u_j w_i \frac{\partial u_i}{\partial x_j} ; \quad P_2 = \mu S^2 ; \quad S = \sqrt{2\nu S_{ij} S_{ij}} ; \]
\[ S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \]  \hspace{1cm} (10a-d)

Further details regarding the transition SST Model are given by Menter et al. (2004 and 2006) and Langtry et al. (2004).

Instead of simulating the whole jet pump, due to the rotational symmetry, only a half of the pump was used as the computational domain. The rotational-symmetry boundary condition was applied for the fluid zone. Hybrid mesh generation was performed with ANSYS Meshing.

For the pump, around 150000 cells were needed to obtain mesh independent solutions. Inflated mesh was used near the wall boundaries to improve accuracy in the boundary layer simulation. In the inflated meshing process, layers of prismatic inflation cells were first generated separately as 5 and 10 layers in the near-wall region. The numerical results obtained using these two layers were then compared with experimental results.

Since the results for the mesh with 5 inflation layers are in slightly better agreement with experimental data and its mesh quality is higher, the inflated mesh was used in the numerical analysis. In 5 layers case, the minimum and average mesh qualities were 8.87% and 82%, respectively. The 3D double-precision pressure-based solver type was selected for a better resolution of the turbulent flow field.

In this numerical simulation study, the total pressures of the motive and entrained fluids are set as the pressure inlet boundary conditions and the static pressure of mixed flow is set as the pressure outlet boundary condition. The values of these boundary conditions are the data measured in experimental tests of the 1/1 scale actual jet pump which is used for validation of CFD results and are listed Table 1.

To specify the turbulence boundary conditions the turbulence intensity and hydraulic diameter were used for all models, and also the intermittency was used for the transition SST model. In the computations for the SST model, the intermittency is set to one (1) at the inlets and outlet of the jet pump. To estimate the values of turbulence intensity, the following formula is employed:
\[ I \approx 0.16 \text{Re}_{D_h}^{-1/8} \]  \hspace{1cm} (11)

For other models, except the transition SST model, the equivalent sand-grain roughness height \( K_\text{s} \) is used to define the wall boundary conditions.

For the same purpose, the geometric roughness height.

<table>
<thead>
<tr>
<th>( P_{pt} ) (Pa)</th>
<th>( P_{pt} ) (Pa)</th>
<th>( P_{pt} ) (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>486829</td>
<td>-11000</td>
<td>123913</td>
</tr>
<tr>
<td>486829</td>
<td>-11000</td>
<td>122946</td>
</tr>
<tr>
<td>486847</td>
<td>-18000</td>
<td>103563</td>
</tr>
<tr>
<td>481851</td>
<td>-27000</td>
<td>84139</td>
</tr>
<tr>
<td>481887</td>
<td>-30500</td>
<td>76424</td>
</tr>
<tr>
<td>482896</td>
<td>-36000</td>
<td>69618</td>
</tr>
<tr>
<td>231793</td>
<td>-18000</td>
<td>29354</td>
</tr>
<tr>
<td>220749</td>
<td>-12500</td>
<td>30403</td>
</tr>
<tr>
<td>202636</td>
<td>-4000</td>
<td>31433</td>
</tr>
<tr>
<td>195614</td>
<td>1500</td>
<td>32489</td>
</tr>
<tr>
<td>187547</td>
<td>7000</td>
<td>33489</td>
</tr>
<tr>
<td>132542</td>
<td>18500</td>
<td>34534</td>
</tr>
<tr>
<td>162426</td>
<td>25000</td>
<td>35601</td>
</tr>
<tr>
<td>162412</td>
<td>24000</td>
<td>33590</td>
</tr>
</tbody>
</table>

K in the transition model is used together with \( K_\text{s} \). However, it is seen that the simulation results did not change in the range of the roughness ratio \( (K_\text{s}/K) \) between 0.4 and 3.0.

The coupled solution algorithm for the pressure-velocity coupling, the least squares cell-based method for the evaluation of gradients and second-order schemes for spatial discretization were selected for this CFD analysis. Thus, all governing and model equations are solved using the second-order schemes. According to the values of the boundary conditions that were entered, the mass flow rates of entrained, motive and mixed fluids were calculated for water jet pump for a given scale and roughness.

When the computations become unstable, the under-relaxation factors were decreased slowly until converged solutions were reached. Numerical solutions were obtained by setting the convergence criteria for all the equations first to \( 10^{-4} \) and then \( 10^{-5} \), and the comparison of the results for these criteria showed that differences between the results were unimportant.

After completing the modeling of flow in the jet pump using the ANSYS Fluent 14.0 code, the effects of scale and roughness on its performance, and some steady characteristics of the flow field were investigated in detail.

### 3. RESULTS AND DISCUSSION

In determining and comparing the performance of all jet pumps (liquid, gas and gas-liquid) dimensionless parameters, which are the ratios of various properties, are used. These parameters for
liquid jet pumps are defined as follows:
The mass flow ratio, the ratio of the entrained fluid mass flow rate to that of motive fluid is given as:

\[ M_r = \frac{\dot{m}_e}{\dot{m}_p} \]  

(12)

The pressure ratio, the ratio of the increase in total pressure of entrained fluid to the decrease in total pressure of motive fluid is given as:

\[ P_r = \left( \frac{p_{se}-p_{st}}{p_{ps}-p_{st}} \right) \]  

(13)

and the jet pump efficiency is found by multiplying these ratios together,

\[ \eta = M_r P_r \]  

(14)

Reynolds number is defined in terms of the hydraulic diameter as:

\[ Re_d = \frac{\rho V D_h}{\mu} \]  

(15)

The scale (geometric) is the ratio of a characteristic length in the model to the corresponding length in the prototype:

\[ S = \frac{L_m}{L_p} \]  

(16)

In this simulation study, the efficiency of 1/1-scale jet pump with an area ratio of 5.92 was first determined numerically as a function of the mass flow ratio using four different turbulence models, namely the realizable k-ε, RSM, SST k-ω and transition SST models. The purpose of this research was to determine which turbulence model provides the best agreement with experimental results. After determining the most appropriate model by comparing with experimental results for the same dimensions and boundary conditions (Yapıcı, 1990), only the best turbulence model was used to investigate the effects of scaling-up and downscaling on the efficiency of the jet pump for seven different scales. Finally, the effects of absolute and relative roughness on the efficiency were analyzed numerically using the full-scale pump.

In order to compare the experimental data with numerical results, the curves for numerical efficiency η and the pressure ratio \( P_r \) are drawn as a function of the mass flow ratio as shown in Fig. 3a. The trends of these characteristic curves indicate that the variations of the dimensionless parameters are similar to those of impeller pumps. The pressure ratio in jet pumps decreases almost linearly with the increase of the mass flow ratio. This is an expected behavior because relatively more energy of the motive fluid is transferred to the entrained fluid for more of the fluid being pumped. However, the experimental and numerical efficiencies first increase up to approximately \( M_r = 1.7 \) and then decrease; that is, there is an optimum range of the mass flow ratio for the models used in current study.

The comparison of the experimental and CFD results for the mass flow ratio at the same pressure ratios is shown in Fig 3b. As seen in this figure, the transition SST model exhibits better results than the other models. While the realizable k-ε and SST k-ω models give results close to the model, the RSM model underestimates the flow ratio and hence the efficiency at higher flow ratios. In other words, the mass flow ratios predicted by the two and four-equation models deviate ±10% from the experimental flow ratios but the predictions of the seven-equation RSM model are slightly out of this range. On the other hand, in the high-efficiency (>30%) operating region in which \( M_r \) ranged from 1.2 to 2.2, the transition SST model provides more accurate predictions. Therefore, the four-equation turbulence model was used in all following simulation studies performed generally in optimum operating conditions.

To interpret why the efficiency of jet pump is lower at mass flow ratios lesser or greater than the optimum mass flow ratio, the flow field within the

![Fig. 3 Comparison of experimental and numerical performances of jet pump (A_r=5.92).](image-url)
pump was viewed by utilizing the outstanding visualization ability of CFD, and the velocity and pressures along its centerline were computed.

For this purpose, as shown in Fig. 4, streamlines at three different flow ratios are plotted in the symmetry plane. As seen in Fig. 4a, larger eddies at the inlet of suction nozzle and smaller eddies near the wall in the entry section of the mixing pipe form at lower mass flow ratios, such as Mr=0.29. The formation of these eddies results in a higher pressure ratio and hence a higher outlet (back) pressure at lower flow ratios. It is observed from the figure that although the jet velocity at the nozzle outlet is very high, it drops suddenly near the middle of the mixing pipe. This is why a larger amount of the energy of the water jet is spent in the eddy formation and thus, less mechanical energy from the liquid is transferred to the entrained water. Moreover, because smaller eddies occur in the outer region between the mixing pipe wall and the jet blowing out of the nozzle, in other words, on the route of the entrained fluid - they will have an influence on decreasing its mass flow rate. These effects reduce the pump efficiency in lower flow ratios. According to Fig. 4b, at the optimum mass flow ratio Mr=1.62, the secondary flow is entrained properly to the mixing pipe without leading to eddies and separations. The jet from the motive nozzle causes the velocity of the entrained water to increase and its velocity decreases gradually along the mixing pipe at the same time. Thus, the kinetic energy of the water jet throughout the pipe is transferred to the secondary fluid without causing additional energy losses and the pump efficiency at the flow ratio reaches a maximum value.

The reason for the drop in efficiency at higher mass flow ratios (Mr=2.44) can be explained as follows. As shown in Fig. 4c, the flow velocity at the mixing pipe outlet or the diffuser inlet is high and the main flow toward the outlet diffuser does not completely fill the cross-section of the pipe. Such a flow generates smaller eddies and therefore, the pressure does not rise sufficiently in the diffuser. In conclusion, a lower pressure ratio and reduced efficiency occur in such operating conditions.

The pressure and velocity variations along the central axis of the jet pump are shown in Fig. 5a and 5b, respectively. Due to a higher back (exit) pressure acting at the lower flow ratio Mr=0.29, the liquid pressure rises sharply slightly ahead of the mixing pipe inlet. This phenomenon results in a drop in the mass flow rate of the entrained fluid.
liquid. Although the pump inlet pressures for the optimum and higher flow ratios differ, their exit pressures are almost the same. The velocity trends along the centerline, as expected, become opposite of the pressure trends, as shown in Fig. 5b.

3.1 Scale effect

Scale effect is an indication of the deviation from the similarity law and varies depending on the geometry of the flow passages. The factor producing this effect in the geometrical similar hydraulic machines is the variation of the friction loss due to the difference in the Reynolds number and the relative roughness of the flow passage wall. Thus, a change in the dissipation of specific hydraulic energy of fluid will directly affect the efficiency of the jet pump.

Physical hydraulic model tests always involve scale effects if S≠1, since it is impossible to correctly model all force ratios. The relevant question is whether or not the scale effects can be neglected (Heller, 2011). One of the research objectives of the current study is a numerical evaluation of the level of scale effects.

The mixing pipe in the experimental jet pump used for comparison had the greatest relative roughness according to its other components and was made of a new drawn steel pipe with $K_r/d_t=4.1\times10^{-6}$. The roughness value was lower than the hydraulically smooth limit of $4.0\times10^{-5}$ and therefore, while investigating the scale effect, the inner surfaces of the pump were assumed as smooth.

In addition to geometric similarity dynamic similarity in liquid jet pumps requires for similarity of model and prototype flows. That is, the model jet pump must satisfy the Reynolds similarity in which the Reynolds numbers of the model and prototype pumps must be the same at the corresponding points. The Reynolds numbers calculated according to Eq. 15 at the various locations of jet pump are shown Table 2 for different scales ranging from 1/4 to 20/1 at the optimum mass flow ratios of the area ratio 5.92. As seen from the table, the Reynolds numbers at inlet of the suction chamber slightly increase with S, but the Re numbers at the other places remain almost the same. When considering a complete jet pump, the range of the Reynolds numbers in the CFD analysis is between $2.78\times10^4$ and $4.9\times10^5$.

In Fig. 6, the efficiency curves are plotted against the pressure ratio for the models in different scales of the full scale (1/1) jet pump, whose the main dimensions are listed in Table 1. As seen from the graph, the maximum value of change in the efficiency with the scale is about 5% in the full range of the flow rate or pressure ratio. The most important result here is that when considering the efficiency of 33.67% of the full scale pump at the optimum $M_r=1.62$, it appeared that the effect of downscaling on the efficiency becomes excessively high when compared to scaling-up. For example, the efficiency at optimum $M_r=1.57$ for $S=1/4$ is 31.36%. The efficiency increases with S and reaches a value of 36.29% at $M_r=1.75$ for $S=20/1$. The efficiency of the jet pump at scales higher than $S=20/1$ remains almost constant.

As is known, jet pumps are being manufactured at different area ratios and hence at different geometries according to the purpose of use of the jet pump. Those jet pumps with higher efficiency have relatively lower flow and area ratios, while the pumps with the higher flow and area ratios have lower efficiencies. The dependence of optimum efficiencies for various scales on area ratios is shown in Fig.7. The boundary conditions and dimensions of the jet pump for the analyzed new area ratios are also the actual values taken from the optimization study performed by Yapici (1990).

According to the numerical simulation results, a maximum efficiency of 38.46% was reached at the optimum area ratio 4.61 and $S=20/1$. In addition to this, the efficiency in the complete range of area ratio rises sharply up to approximately $S=3$, but for $S>3$ the increase rate of the efficiency decreases and then the efficiency tends to remain unchanged. The reason for such a trend can be explained by Fig. 8. It can be seen in Fig. 8a that the peak value for the rate of dissipation of the area-averaged turbulent kinetic energy and the area between the constant scale curve S and horizontal axis-X diminish very sharply with the increase of the scale S. All the peaks arise in a narrow region between the motive nozzle outlet and the suction nozzle inlet where the high-velocity jet meets with the low-velocity suction flow and the velocity gradient in the radial direction has its highest value.

For example, the peak value of the energy dissipation, which is 3134 W/kg for 1/4 scale, drops below 50 W/kg for the 10/1 scale. Thus, the increase of the rate of energy dissipation with decreasing scale results in a considerable reduction in the efficiency of jet pump.

Similar behavior is clearly exhibited in Fig. 8b, which is drawn by using the volume-averaged value representing more accurately the dissipation rate of the energy. The figure gives the simulation
Table 2  Reynolds number intervals at various locations of jet pump for scales from 1/4 to 20/1.

<table>
<thead>
<tr>
<th>Location</th>
<th>$Re_p/10^4$</th>
<th>$Re_{mp/m}/10^5$</th>
<th>$Re_s/10^4$</th>
<th>$Re_d/10^5$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Motive nozzle inlet</td>
<td>7.28 -7.34</td>
<td></td>
<td>3.85-4.09</td>
<td></td>
</tr>
<tr>
<td>Middle of mixing pipe</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Suction chamber inlet</td>
<td></td>
<td></td>
<td>2.78-3.53</td>
<td></td>
</tr>
<tr>
<td>Diffuser outlet</td>
<td></td>
<td></td>
<td>1.63-1.71</td>
<td></td>
</tr>
</tbody>
</table>

Fig. 6  Variations of efficiency with pressure ratio for different scales.

Fig. 7  Optimum efficiency dependence on area ratio for different scales.

results for jet pumps with rough and smooth wall surfaces. While the dissipation rate of the energy for $S<1/1$ depends very strongly on the scale $S$, for $1<S<5$ its dependence on $S$ attenuates highly and for $S>5$ it almost remains constant. As a result, the reduction of the energy dissipation rate with increasing $S$ at lower scales results in the increase of the pump efficiency, but the dissipation rate remains almost constant together with the dissipation rate at the higher scales, regardless of the roughness size.

3.2 Surface roughness effect

In this stage of study, simulations of 1/1 scale jet pump were performed for four different roughness sizes, in order to investigate the influence of the surface roughness on efficiency. The numerical simulation results are shown as a function of the mass flow ratio in Fig. 9. The pump efficiency increases considerably with the decrease in the roughness height until the hydraulically smooth limit was reached. Essentially, a higher roughness height means a lower mass flow ratio and hence, lower efficiency. If such a jet pump is manufactured from stainless steel with a roughness height of 0.002 mm or plastic material, instead of cast iron with a roughness height of 0.26 mm, the optimum efficiency rises from 26.18% to 33.60%. In this case, the performance increment ratio can be about 28%. Moreover, according to these results, to reach optimum efficiency at a given area ratio, it will be sufficient to use a hydraulically smooth pipe or material in the manufacturing of a jet pump.

Fig. 8  Variations in rate of dissipation of turbulent kinetic energy ($Ar=5.92$).
For the purpose of comparison, the curves showing influence of absolute and relative roughness on pump efficiency are shown in Fig. 9. Effect of surface roughness on efficiency.

When performing simulations in the range of large relative roughness, very slightly different results were encountered, as shown in Fig. 11. At both the lower scale and relative roughness, the efficiency varies with S, in contrast to its normal trend. That is, if these dimensionless parameters (S and K/d) are both reduced, the value of efficiency becomes different from the one which is independent of S for a given K/d.

Fig. 10 for A_r=5.92. After the efficiency at the constant absolute roughness K_s=0.1 increases up to about S=20, it remains constant at a value of 34%. The same conclusion was previously obtained for K_s=0 using six different areas and nine different scales and the numerical simulation results are demonstrated in Fig. 7. Since the relative roughness decreases when increasing the scale while maintaining constant the absolute roughness height, the friction loss coefficients of the components of the jet pump will become smaller, in the first instance in the mixing pipe. Thus, the reduction of the hydraulic energy losses occurring in the pump flow passages does have an effect on the increase of the efficiency. However, if the scale is increased at a constant relative roughness, the efficiency remains approximately constant, although there are small changes at the reduced scales. Clearly this shows that the Reynolds similarity together with complete geometrical similarity is satisfied and hence the efficiency of water jet pump is independent of the scale.

Fig. 10 Comparison of effect of absolute and relative roughness on optimum efficiency.

Fig. 11 Optimum efficiencies for various relative roughness values.

Fig. 12 Relative efficiency versus relative roughness (A_r=5.92).

Fig. 12 shows the variations of the optimum efficiency and mass flow ratio with the relative roughness of the mixing pipe for full scale (1/1) pump. The optimum efficiency ratio (or relative efficiency) is defined as the ratio of the optimum efficiency of the jet pump in a given roughness size to that of a pump assumed hydraulically smooth (Yamazaki et al., 2006): $\eta_{opt.r} = \eta_{opt}/\eta_{opt,hs}$. It is concluded from this figure that as the relative roughness increases, both the mass flow and optimum efficiency ratio decrease, but more quickly at lower relative roughness values. If it is considered that the relative efficiency at the relative roughness value of 0.05 is 60%, the importance of the roughness is better comprehended. Since the CFD simulations can capture the effect of a parameter with very small physical size such as surface roughness, it appears
to be a very good tool to use in the design of jet pumps to improve their efficiency.

4. CONCLUSIONS

In this simulation study, the numerical performance data for a water jet pump with the area ratio of 5.92 and the relative mixing pipe length of 7.4 are computed using the four different turbulence models, namely the realizable $k-\varepsilon$, RSM, SST $k-\omega$ and transition SST model. The numerical results of the models were compared with the previous experimental results for the same pump and it was determined that the transition SST model produced predictions closer to the actual data than other models. Using the model, the effects of roughness on the optimum pump efficiency were investigated in a scale range between 1/4 and 20/1 for $Ar=5.92$. Also the scale effects on the efficiency of six jet pumps with different area ratios were investigated while the roughness remained constant.

The scale effect in the jet pump with a given absolute roughness size is very important up to a given scale size. At a constant relative roughness, the efficiency of the pumps does not depend on the scale size, except for the lower relative roughness and scale size. If the area ratio and the scale are constant, the efficiency is greatly affected by the relative roughness. Therefore, in the design of liquid jet pumps and the model studies, the scale and roughness effects should be considered especially if these pumps are to be used in large-scale pumping systems.

One of the important numerical simulation results is that a maximum efficiency of 38.46% is achieved at the optimum area ratio of 4.61 and $S=20/1$ for a hydraulically smooth jet pump, and for $S>20$ the efficiency remains almost the same. Moreover, this study shows that CFD simulations can be used to improve the efficiency of liquid jet pumps.

ACKNOWLEDGEMENT

This research was supported by Selçuk University under the BAP Project No: 11201037.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_r$</td>
<td>Area ratio = $A_{tr}/A_{tn}$ (mixing chamber area/nozzle outlet area)</td>
</tr>
<tr>
<td>$d$</td>
<td>Diameter (m)</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulent kinetic energy ($m^2/s^2$)</td>
</tr>
<tr>
<td>$K$</td>
<td>Roughness height (mm)</td>
</tr>
<tr>
<td>$L$</td>
<td>Length (m)</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass flow rate (kg/s)</td>
</tr>
</tbody>
</table>

$M_r$ Flow rate ratio ($m_{d}/m_{p}$)

$p$ Static pressure (kPa)

$P_r$ Pressure ratio

$Re$ Reynolds number

$S$ Scale

$u_i$ Time-averaged velocity (m/s)

$u'$ Fluctuation velocity component (m/S)

$x_i$ Coordinate (m)

Greek letters

$\varepsilon$ Turbulent kinetic energy dissipation ($m^2/s^3$)

$\eta$ Jet pump efficiency

$\mu$ Dynamic viscosity (Pa.s)

$\mu_t$ Turbulent viscosity (Pa.s)

$\rho$ Fluid density (kg/m$^3$)

$v$ Kinematic viscosity (m$^2$/s)

$\omega$ Specific turbulent dissipation rate ($s^{-1}$)

Subscripts

$d$ discharge/outlet

$n$ nozzle

$p$ motive/primary fluid

$t$ mixing pipe, total

$s$ suction, smooth, sand

REFERENCES


