Study of Boundary Conditions in Single-Blade Pump Simulation

AUTHOR(S)
Juhaveikko Ala-Juusela

ABSTRACT
Single-blade sewage pump is simulated numerically using FINFLO flow solver. Low-Reynolds number approach is utilized with a $k-\epsilon$ turbulence closure. The effect of the boundary conditions on a performance prediction, mass balance fluctuation and velocity distributions is studied.

MAIN RESULT
Pump head, efficiency and torque are obtained from the simulated flowfield and compared to measured data. The velocity distributions are also compared to the measurements available. Reversed in- and outflow boundary conditions do not remove the fluctuation in mass balance, it only changes location. Modified impeller surface boundary condition has a small effect on pump performance but more visible on velocity distributions.

KEY WORDS
FINFLO, $k-\epsilon$ model, pump performance

APPROVED BY
Timo Siikonen    30th May 2007
Contents

Nomenclature 3

1 Introduction 5

2 Flow Equations 5

3 Turbulence Modelling 7

4 Computational Grids and Boundary Conditions 8

5 Results 9

6 Discussion 12

A Convergence 14

B Pump performance with 200 internal iterations in all simulations 15

C Distributions 16
Nomenclature

\( C_p \) pressure coefficient
\( E \) total energy
\( F, G, H \) flux vectors in the \( x-, y- \) and \( z- \) directions
\( H \) total head
\( Pr \) Prandtl number
\( Q \) source term vector
\( Re \) Reynolds number
\( T \) temperature
\( U \) vector of conservative variables
\( V \) velocity vector
\( c_p \) specific heat at a constant pressure
\( c_v \) specific heat at a constant volume
\( e \) specific internal energy
\( k \) turbulent kinetic energy
\( \dot{m} \) mass flow
\( p \) pressure
\( \dot{q} \) heat flux
\( t \) time
\( u, v, w \) velocity components in the \( x-, y- \) and \( z- \) directions
\( u_\tau \) friction velocity (:math:`\sqrt{\tau_w/\rho}`)
\( y^+ \) dimensionless distance from the wall (:math:`y u_\tau/\nu`)
\( \beta \) artificial compressibility coefficient
\( \delta_{ij} \) Kronecker’s delta
\( \epsilon \) dissipation of the kinetic energy of the turbulence
\( \theta \) temperature difference (:math:`T - T_\infty`)
\( \mu \) dynamic viscosity
\( \nu \) kinematic viscosity
\( \phi \) scalar
\( \rho \) density
\( \tau \) shear stress

Superscripts

\( T \) transposition
\( l \) left side
\( r \) right side
\( w \) wall value
\( ' \) fluctuating component

Subscripts

\( T \) turbulent
\( i, j, k \) \( i-, j-, k- \) component
\( t \) tangential component
\( n \) normal component
1 Introduction

In this study a single-blade sewage pump is simulated numerically using FINFLO flow solver. A low-Reynolds number $k - \epsilon$ turbulence model is utilized in the simulations. The case has been simulated earlier using the velocity as an inlet boundary condition and the pressure as an outlet boundary condition [1]. Since this set of boundary conditions caused severe oscillations in the mass flow, in this study the pressure is given as an inlet boundary condition and a constant velocity distribution as an outlet boundary condition. Furthermore, the effect of the size of the rotating surface of the impeller is studied. In order to simplify the grid structure the hub and the shroud have a same diameter, which makes the impeller surface larger than in reality. This is corrected by changing a part of boundary condition of the impeller from a rotating surface to solid. However, it is still a part of the rotating grid block.

In the following, the governing equations and turbulence modelling are firstly described. Next, the computational domain and the grid are depicted and, finally, the results of the simulation are presented and compared to measurements as well as previous simulations with the different boundary condition approach.

2 Flow Equations

A low-Reynolds number approach is used in FINFLO. The Reynolds-averaged Navier-Stokes equations, and the equations for the kinetic energy ($k$) and dissipation ($\epsilon$) of turbulence can be written in the following form

$$\frac{\partial U}{\partial t} + \frac{\partial (F - F_t)}{\partial x} + \frac{\partial (G - G_t)}{\partial y} + \frac{\partial (H - H_t)}{\partial z} = Q \tag{1}$$

where the unknowns are $U = (\rho, \rho u, \rho v, \rho w, E, \rho k, \rho \epsilon)^T$. The inviscid fluxes are

$$F = \begin{pmatrix} \rho u \\ \rho u^2 + p + \frac{2}{3} \rho k \\ \rho v \\ \rho v u \\ \rho w \\ (E + p + \frac{2}{3} \rho k) u \\ \rho u \epsilon \end{pmatrix}, \quad G = \begin{pmatrix} \rho w \\ \rho w v \\ \rho w^2 + p + \frac{2}{3} \rho k \\ \rho w u \\ \rho w v \\ (E + p + \frac{2}{3} \rho k) v \\ \rho w \epsilon \end{pmatrix}, \quad H = \begin{pmatrix} \rho w \\ \rho w w \\ \rho w v \\ \rho w^2 + p + \frac{2}{3} \rho k \\ \rho w k \\ (E + p + \frac{2}{3} \rho k) w \\ \rho w \epsilon \end{pmatrix} \tag{2}$$

where $\rho$ is the density, the velocity vector by using Cartesian components is $\vec{V} = u \hat{i} + v \hat{j} + w \hat{k}$, $p$ is the pressure, $k$ is the turbulent kinetic energy and $\epsilon$ its dissipation, and the total energy $E$ is defined as

$$E = \rho e + \frac{\rho \vec{V} \cdot \vec{V}}{2} + \rho k \tag{3}$$
where \( c \) is the specific internal energy. The viscous fluxes are

\[
F_v = \begin{pmatrix}
0 \\
\tau_{xx} \\
\tau_{xy} \\
\tau_{xz} \\
u \tau_{xx} + v \tau_{xy} + w \tau_{xz} - q_x \\
\mu_k \frac{\partial k}{\partial x} \\
\mu_e \frac{\partial \varepsilon}{\partial x}
\end{pmatrix}
\]

\[
G_v = \begin{pmatrix}
0 \\
\tau_{xy} \\
\tau_{yy} \\
\tau_{yz} \\
u \tau_{xy} + v \tau_{yy} + w \tau_{yz} - q_y \\
\mu_k \frac{\partial k}{\partial y} \\
\mu_e \frac{\partial \varepsilon}{\partial y}
\end{pmatrix}
\]

\[
H_v = \begin{pmatrix}
0 \\
\tau_{xz} \\
\tau_{yz} \\
\tau_{zz} \\
u \tau_{xz} + v \tau_{yz} + w \tau_{zz} - q_z \\
\mu_k \frac{\partial k}{\partial z} \\
\mu_e \frac{\partial \varepsilon}{\partial z}
\end{pmatrix}
\]

Here the stress tensor, \( \tau_{ij} \), includes laminar and turbulent components. The fluid is assumed to be Newtonian and, therefore, the laminar stresses are modelled by using Stokes hypothesis. The Reynolds stresses \( \rho u_i^l u_j^T \) are included in the stress tensor \( \tau_{ij} \).

\[
\tau_{ij} = \mu \left[ \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} - \frac{2}{3} (\nabla \cdot \bar{V}) \delta_{ij} \right] - \frac{\rho u_i^l u_j^T}{3} + \frac{2}{3} \rho k \delta_{ij}
\]

(5)

For the Reynolds stresses, Boussinesq’s approximation

\[
-\rho u_i^l u_j^T = \mu_T \left[ \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} - \frac{2}{3} (\nabla \cdot \bar{V}) \delta_{ij} \right] - \frac{2}{3} \rho k \delta_{ij}
\]

(6)

is utilized. Here \( \mu_T \) is a turbulent viscosity coefficient, which is calculated by using a turbulence model, and \( \delta_{ij} \) is the Kronecker’s delta. In the momentum and energy equations, the kinetic energy contribution \( 2/3 \rho k \delta_{ij} \) has been connected with pressure and appears in the convective fluxes, whereas the diffusive part is connected with the viscous fluxes. The viscous stresses contains a laminar and a turbulent parts. The heat flux can be written as

\[
\bar{q} = -(\lambda + \lambda_T) \nabla T = - \left( \mu + \frac{\mu_T}{\sigma_k} \right) \frac{c_p}{P_r} \nabla T
\]

(7)

where \( \lambda \) is a molecular and \( \lambda_T \) a turbulent thermal conductivity coefficient and \( P_r \) is a laminar and \( P_r_T \) a turbulent Prandtl number, and \( c_p \) is a specific heat at constant pressure. The diffusion of turbulence variables is modelled as

\[
\mu_k \nabla k = \left( \mu + \frac{\mu_T}{\sigma_k} \right) \nabla k
\]

(8)

\[
\mu_e \nabla \varepsilon = \left( \mu + \frac{\mu_T}{\sigma_\varepsilon} \right) \nabla \varepsilon
\]

(9)
where $\sigma_k$ and $\sigma_\epsilon$ are turbulent Schmidt’s numbers of $k$ and $\epsilon$, respectively. Density is obtained from an equation of state $p = p(\rho, T)$ Since this case is essentially incompressible, pressure differences $p - p_0$ are solved instead of pressure. The components of the source term $Q$ are non-zero in possible buoyancy terms and in turbulence model equations. In this study the buoyancy terms are insignificant and not applied.

In the present study an artificial compressibility approach is used to determine the pressure. The flux calculation is a simplified version of the approximate Riemann-solver utilized for compressible flows [2]. It should be noted that in this approach the artificial sound speed affects the solution, but the effect is of the second-order and it should not be visible as the grid is refined. The effect is similar to the Rhie and Chow interpolation method applied in commercial codes. The solution method applied is described in [3]. In time-accurate simulations the artificial compressibility approach is used inside a physical time step [4]. Each time step is treated as a steady-state case and iterations are made inside the time step. The time derivative term is treated as a source term. The method is fully implicit and a three-level approximation is used for the time derivative.

### 3 Turbulence Modelling

As mentioned, turbulent stresses resulting from the Reynolds averaging of the momentum equation are modelled by using Boussinesq’s approximation (6). The turbulent viscosity coefficient $\mu_T$ is determined by using Chien’s [5] low-Reynolds number $k - \epsilon$ model from the formula

$$\mu_T = c_\mu \frac{k^2}{\epsilon}$$  \hspace{1cm} (10)

where $c_\mu$ is a empirical coefficient. The source term of Chien’s model is

$$Q = \left( \begin{array}{c} P - \rho \epsilon - 2 \mu \frac{k}{y^+_n} \\ c_1 \frac{\epsilon}{k} P - c_2 \frac{\beta^2}{k} - 2 \mu \frac{\epsilon}{y^+_n} e^{-y^+ / 2} \end{array} \right)$$  \hspace{1cm} (11)

where $y_n$ is the normal distance from the wall, and the dimensionless distance $y^+$ is defined by

$$y^+ = y_n \frac{\rho u_\tau}{\mu} = y_n \sqrt{\frac{\rho \tau_w}{\mu}} \approx y_n \left[ \frac{\rho |\nabla \cdot \mathbf{V}|}{\mu} \right]_{w}^{1/2}$$  \hspace{1cm} (12)

Here $u_\tau$ is friction velocity and $\tau_w$ is friction on the wall, and the connection between them is $u_\tau = \sqrt{\tau_w / \rho}$. The unknown production of the turbulent kinetic energy is modelled using Boussinesq’s approximation (6)

$$P = -\rho u_i u_j \frac{\partial u_i}{\partial x_j}$$

$$= \left[ \mu_T \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) - \frac{2}{3} \delta_{ij} \rho k \right] \frac{\partial u_i}{\partial x_j}$$  \hspace{1cm} (13)
The turbulence model presented above contains empirical coefficients. Those are given by [2]

\[
\begin{align*}
  c_1 &= 1.44 \\
  c_2 &= 1.92(1 - 0.22e^{-Re_T^2/36}) \quad \sigma_k = 1.0 \\
  c_\mu &= 0.09(1 - e^{-0.015u^+}) \quad \sigma_\epsilon = 1.3
\end{align*}
\]

where the turbulence Reynolds number is defined as

\[
Re_T = \frac{\rho k^2}{\mu \epsilon}
\]

Chien’s model is very robust, but it has several shortcomings. It usually overestimates the turbulence level and is not performing well in a case of an increasing pressure gradient.

4 Computational Grids and Boundary Conditions

The computational grid is described in detail in reference [1]. Only the second grid level (every other gridpoint) is used in this simulation, since the main purpose is to study the effect of boundary conditions and a grid converged results are not important. The surface grid is depict in Fig. 1. The area where the boundary condition is changed from rotating surface to solid is shown using red color.

**Fig. 1:** Surface grid of the impeller, red area demonstrates the area, where boundary conditions are changed from a rotating surface to a solid surface.
The pressure, temperature and turbulence quantities are given as an inlet boundary condition. The normal velocity is extrapolated from the computational domain. A value of 800 000 Pa is given for the inlet pressure in order to prevent negative pressures during the iteration. The pressure level is insignificant in a present incompressible case. For the outlet boundary condition the normal velocity is given and the outlet pressure, as well as the other variables are extrapolated from the flowfield.

5 Results

The case where the whole impeller grid surface is rotating is referred as an original geometry, and the case with a partly solid boundary condition is referred as a modified geometry. The results from the earlier study [1] are referred as a pressure outlet case.

A quasi-steady simulation was used as as an initial guess for the time-accurate simulation, where twelve impeller rounds were simulated. The modified geometry is used only for the last eight impeller rounds of the time-accurate simulation. In the both simulations, 200 internal iterations are used. The Courant number of the internal iterations is one in all simulations. In the earlier pressure outlet simulation, 135 internal internal iterations were used.

Multigrid could not be used to accelerate convergence in the time-accurate simulations due to inexplicable instability, in the quasi-stedy computations two multi-grid layers were used. Some convergence histories within a timestep are shown in Figs. A-1 – A-3 in Appendix A. It is seen that the boundary type does not affect the convergence properties. The convergence of the shaft power is slow and it is questionable is the number of iteration cycles sufficient.

A head is calculated from

$$H = \frac{p_2 - p_1}{\rho g}$$

(16)

where $p_1$ is the inlet pressure, $p_2$ the outlet pressure; $\rho$ the density, and $g$ the acceleration due to gravity.

The efficiency is obtained from

$$\eta = \frac{\Delta E}{T \omega}$$

(17)

where $\Delta E$ is the difference between the mechanical energy flux at the inlet and the outlet and $T \omega$ is the required axial power. The axial power should be smaller than the measured one, since several components of the loss, for example the friction on the outside of the impeller and bearing losses, are not taken into account. The head, efficiency and axial power needed are presented in Table 1.
The head, mass balance, efficiency and axial power over a one impeller cycle are shown in Figs. 2 and 3. In the figures there are also results from the pressure outlet simulation. Internal iterations in this simulation is 135. Similar figures but with 200 internal iterations in all simulations are shown in Figs. B-1 and B-2 in Appendix B. It can be seen, that both the original geometry and the modified geometry case are really close to each other. The curves of the head, the massflow and the efficiency oscillate more than once per a cycle. This is caused by a reflection from the outlet. In the pressure outlet case there is no reflection when 135 internal iterations are used, but when the number of iterations is increased to 200, oscillations are even bigger than in present cases (see Appendix B).

![Graph](image)

**Fig. 2:** Head (left) and massflow (right) as a function of the impeller angle in the time-accurate simulations with the original and the modified geometries as well as the pressure outlet case.
Fig. 3: The efficiency (left) and the shaft power (right) as a function of the impeller angle in the time-accurate simulations with the original and the modified geometries as well as the pressure outlet case.

Pressure distributions on the surface of the impeller and on a half of the volute, as well as velocity vectors and velocity distributions at the measurement stations, given in Fig. 4, are shown in Figs. C-1 – C-8 of Appendix C. In measurement station 1, the top part of the distribution is smoother when the modified impeller is used, but otherwise the distributions are similar. Pressure outlet case is clearly different from the present cases, starting from the pressure distribution, which oscillates in a different pace. Also velocity distributions are more similar between the present cases than between pressure outlet and velocity outlet cases with same impeller geometry.

Fig. 4: Locations of the measurement points and directions of the tangential and radial velocities.
Distributions for the velocity and turbulent kinetic energy, taken on a plane at a distance of 40 mm from the bottom of the impeller, are shown in Figs. C-9 - C-10. The velocity in the figures is in an inertial coordinate system. In the velocity distributions there are only small differences, but in the turbulent kinetic energy distributions the shape of the contours in the outlet pipe is biased to the left side of the pipe in the modified case, while in the original impeller case there are two equal size peaks.

6 Discussion

In this study a single-blade sewage pump is simulated numerically using a low-Reynolds number $k - \varepsilon$ turbulence closure. The pressure is given as an inlet boundary condition and a constant velocity distribution as an outlet boundary condition. This is opposite to the previous studies where the pressure is defined at the outlet and the velocity at the inlet. Furthermore, the effect of the size of the rotating surface of the impeller is studied by changing a part of boundary condition of the impeller from rotating to solid.

The goal of the study was to examine, whether utilizing the velocity as the outlet condition would prevent error in the mass balance and also reflections from the outlet, which occur as unphysical pressure fluctuations. The mass flow error does not vanish by changing the fixed velocity boundary from one end to another. The fluctuations of the mass flow just change place from the outlet to the inlet. Pressure fluctuations diminish significantly comparing to the pressure outlet case, when the same amount of internal iterations is used. On the other hand the pressure oscillations are much smaller in the pressure outlet case, when the number of internal iterations is smaller. The role of the number of internal iterations was not clearly understood when the simulation was made. It seems that there is at least in this case a local maximum when considering oscillations and, therefore, finding optimal number of internal iterations might be difficult. Both smaller and greater number of iterations should be tried, before confident conclusions can be made.

The modification in the impeller boundary condition affects clearly to the velocity distributions. The difference is small, and does not affect the impeller-volute interaction. Even the pump performance is affected only slightly by this modification.
References


A Convergence

Fig. A-1: Convergence history of the $\|L\|_2$-norm of the $x$-momentum residual within internal simulations (one timestep), original geometry on the left and modified on the right.

Fig. A-2: Convergence history of the budget of the turbulent energy within internal simulations (one timestep), orientation as in Fig. A-1.

Fig. A-3: Convergence history of the shaft power within internal simulations (one timestep), orientation as in Fig. A-1.
B  Pump performance with 200 internal iterations in all simulations

Fig. B-1: Head (left) and massflow (right) as a function of the impeller angle in the time-accurate simulations with the velocity outlet boundary condition using the original and modified geometries as well as the simulation with the pressure outlet boundary condition.

Fig. B-2: The efficiency (left) and the shaft power (right) as a function of the impeller angle in the time-accurate simulations with the velocity outlet boundary condition using the original and modified geometries as well as the simulation with the pressure outlet boundary condition.
C Distributions

Fig. C-1: Pressure distribution on the surface of the impeller and volute, velocity vectors and velocity distributions at measurement point 1. From top to bottom: The original geometry, the modified geometry and the pressure outlet case. The angle is 0°.
Fig. C-2: Pressure distribution on the surface of the impeller and volute, velocity vectors and velocity distributions at measurement point 1. The orientation as in Fig. C-1. The angle is 90°.
Fig. C-3: Pressure distribution on the surface of the impeller and volute, velocity vectors and velocity distributions at measurement point 1. The orientation as in Fig. C-1. The angle is 180°.
Fig. C-4: Pressure distribution on the surface of the impeller and volute, velocity vectors and velocity distributions at measurement point 1. The orientation as in Fig. C-1. The angle is 270°.
Fig. C-5: Pressure distribution on the surface of the impeller and volute, velocity vectors and velocity distributions at measurement point 2. The orientation as in Fig. C-1. The angle is 0°.
Fig. C-6: Pressure distribution on the surface of the impeller and volute, velocity vectors and velocity distributions at measurement point 2. The orientation as in Fig. C-1. The angle is 90°.
Fig. C-7: Pressure distribution on the surface of the impeller and volute, velocity vectors and velocity distributions at measurement point 2. The orientation as in Fig. C-1. The angle is 180°.
Fig. C-8: Pressure distribution on the surface of the impeller and volute, velocity vectors and velocity distributions at measurement point 2. The orientation as in Fig. C-1. The angle is 270°.
Fig. C-9: Velocity distribution at plane 40 mm from the bottom of the impeller. The original geometry results on the left and the modified geometry results on the right, from top to bottom: 0°, 90°, 180° and 270°.
Fig. C-10: Distribution of the turbulent kinetic energy at plane 40 mm from the bottom of the impeller. The orientation same as in Fig. C-9.