CFD modelling of a jet pump with circumferential nozzles for large flow rates

E. Lisowski a,*, H. Momeni b

a Institute of Applied Informatics, Cracow University of Technology, al. Jana Pawła 37, 31-864 Kraków, Poland
b Institute of Machine and Marine Technology, Bergen University College, Nygårdsgaten 112, N-5020 Bergen, Norway

* Corresponding author. E-mail address: lisowski@mech.pk.edu.pl

Received 30.04.2010; accepted in revised form 30.05.2010

Abstract

Jet pumps are devices which are widely used in transport of dangerous or aggressive goods, contaminations like solid particles or even food. The principle of jet pumps which is transferring energy of one fluid that is called primary or motive is transferred into a secondary fluid makes that such pumps have no moving parts and are able to work in remote in extreme conditions. Jet pumps main disadvantageous is low efficiency ranging 40 % and low pumping height which significantly limits its usage. One of the way to eliminate the low pumping height is to applying second stage of motive fluid. The other way is correct design of motive nozzles. Presented in the paper jet pump is a Liquid Jet Liquid (LJL) class of jet pump which both, motive and secondary fluids are water. This pump is a two stage jet pump with circumferential nozzles which inlet port has diameter approx. 200 mm and is assigned for high flow rate up to 200 m³/h. The main aim undertaken in this paper is to investigate flow phenomena that appears during jet pump operation as well as investigating influence of various design of motive nozzles on value of head pressure. Investigated jet pump is a prototype that was prepared in cooperation between Bergen University College and Fjell Industries AS, Norway. Modelling of jet pump is a complex task and makes difficulties in modelling when using traditional approach therefore to simulate phenomena that appears in jet pump during operation was used CFD tool: Ansys CFX code. Numerical simulations allowed to obtain information about pressure and velocity distribution as well as estimate the head pressure for various design of motive nozzle. There was investigated three design of motive nozzle: standard, with additional circumferential holes and with set of circumferential holes on motive nozzle. Conducted research shown that by modification of motive nozzle is possible to increase pumping height with almost 45%.

Keywords: modelling, simulation, flow simulation, cfd analysis, jet pump

1. Introduction

Jet pumps are a pumping devices that transfer energy from one fluid which is called motive or primary to another one (called driven or secondary). Motive fluid might be liquid or a gas as well, while secondary might be liquid, gas or a mixture. It might be found different names of jet pumps depending on theirs applications [1]. They are called ejectors, injectors, eductors, elevators, gas compressors or exhauster when applied to gases. In principle they are the same family of pumps that transforming one energy from one fluid to another [2]. The schema of jet pump is presented in fig. 1. It might be found a nozzle, where motive fluid flows into the pump, entertainment where motive and driven fluids meets each other, throat where fluids are mixing and finally discharge where both fluids leaves the pump.
Jet pumps have a lot of advantageous, which the most important is no moving parts resistance for wear and lower production cost in comparison to other types of pump. These caused a wide applications of jet pumps to transport hazardous liquids, rocks with limited dimensions, slurries or even food or naval resources like fish and shellfish. The disadvantageous of jet pumps is low efficiency and height of pumping. It might be distinguished two main types of pump (fig. 2 and 3) that are differ with nozzle of motive fluid which might be axial or circumferential. The nozzle (1) accelerates motive fluid to high velocity. Kinetic energy and momentum of motive fluid is gained at the cost of pressure energy. Pressure falls in mixing chamber (3) and secondary fluid is forced to flow into the motive fluid stream through the inlet (2). Mixed fluids than go to the discharge chamber (4), where fluids deaccelerate and gain pressure energy.

The main advantageous of circumferential nozzle is ability to transport non liquid objects with limited dimensions. The area of application for jet pumps is very wide, in chemical and oil industry for nuclear engineering. There might be a lot of solutions for jet pumps. Jet pump delivery and in consequences its dimensions may range from few millimeters to few meters.

### 2. The object of modelling

The object of the work is a two stage liquid jet liquid (LJL) pump that is shown in fig. 4. It is a prototype of two stage pump that was elaborated in cooperation between Bergen University College and Fjell Industries AS. It is a jet pump with circumferential motive nozzles with suction diameters 200 mm what significantly extends applications of this pump on a wide range of liquids and mixtures. The pump consist of suction port (1), outlet port (2), stage one high pressure port (3), stage two high pressure port (4), circumferential nozzle of stage one (5) and circumferential nozzle of stage two (6).

The pump presented on above figure was modified to increase pumping height as much as possible. Therefore there was investigated two solutions of LJL pump which differ in design of circumferential nozzle of stage one. Below figure shows pump with nozzle with additional circumferential holes at motive nozzle.

The next step of modifications was increasing number of holes on motive nozzle what is shown in fig. 6.
3. Mathematical model

Mathematical model of jet pump consist of mass, momentum and energy conservation equations. When we assume a fixed in space volume $\Omega$, bounded by close surface $S$, the mass conservation equation in general form is the following (according to [3]):

$$\frac{\partial}{\partial t} \int_\Omega \rho d\Omega + \oint_S \rho \mathbf{v} \cdot dS = 0$$

where: $\rho$-fluid density, $\mathbf{v}$-fluid velocity, $t$-time.

The momentum conservation equation in general form is as follows:

$$\frac{\partial}{\partial t} \int_\Omega \rho \mathbf{v} d\Omega + \oint_S \rho \mathbf{v} (\mathbf{v} \cdot dS) =$$

$$\oint_S p \cdot dS + \oint_S \tau \cdot dS + \oint_S \rho f_e d\Omega$$

where: $\tau$-shear stress tensor, $f_e$-vector of external forces, $p$-pressure.

The energy conservation equation in general form is as follows:

$$\frac{\partial}{\partial t} \int_\Omega E d\Omega + \oint_S (\rho H - k \nabla T - \tau \cdot \mathbf{v}) \cdot dS =$$

$$\oint_S (\rho f_e \cdot \mathbf{v} + q_H) d\Omega$$

where: $E = e + \frac{v^2}{2}$-total energy, $e$-internal energy, $H = E + \frac{p}{\rho}$-is enthalpy, $W_f = \rho f_e \cdot \mathbf{v}$-is a work of external forces, $q_H$-is heat released by chemical reactions, $k$-is conductivity coefficient, $T$-is absolute temperature.

Research on jet pumps mainly theoretical [4, 5, 6, 7, 8] are focused on defining pumps parameters like dimensions of nozzles, angles of diffusers, etc. There are also conducted semi empirical research [9] or typically empirical [10]. All of them do not allow to modeling phenomena that appears during flow accurate enough. Developed within last years computer simulation tools like Computational Fluid Dynamics gives much wider possibilities in modeling fluid flow is also found in modeling of jet pumps [11].

4. CFD analysis

CFD analysis was carried out for steady state conditions with variable flow rate at jet pump motive inlet with the following assumptions:

1. Flow is incompressible and homogenous.
2. There is no heat transfer between fluids and environment.
3. Roughness of walls was neglected.
4. Analysis was conducted for steady state conditions.

5. CFD results

5.1. It was assumed the following fluid properties: density $\rho=1000$ [kg/m³], viscosity $\mu=1.003 \times 10^{-3}$ [Pa s].
5.2. The flow is turbulent, a Reynolds Stress Model was assumed.
5.3. Standard wall function was used.

The CFD simulations deals with case when only one stage is operating. Due to the symmetry of geometrical model, only a half was used, the remaining was replaced by symmetry boundary conditions (fig. 7).

Before the CFD model was prepared the grid was made in Ansys CFX mesh tool. The grid was prepared with mixed tetrahedral and prism cells. Prismatic cells were applied in the vicinity the wall. fig. 8 and 9 present a grid for a jet pump.
distribution are shown on fig. 10 and 11. These are exemplary results for the case when flow rate at the stage one motive flow rate is 15 dm³/s.

Fig. 10. Velocity distribution (in m/s)

Fig. 10. Pressure distribution (in Pa)

CFD simulations was also used to assessing influence of modifications presented in fig. 4, 5, 6 on a head pressure of jet pump. Below figure shows an influence of modification of jet pump on head pressure. What can be noticed is that using additional radial holes on motive nozzle allows significantly increase pumping height, which in comparison for nozzle without additional holes is 45% higher.

6. Conclusions

Presented in this paper CFD simulations allowed to obtain information about flow phenomena appearing during pump operation as well as assess an influence of modification of motive nozzle on head pressure. Presented in the paper two solutions of jet pump motive nozzle significantly allowed to increase pumping height up to 45%. Used CFD tool Ansys CFX code appeared to be an efficient tool in modeling of jet pump. Presented modification in the paper shown that with modifications of nozzle geometry is possible to increasing pump head pressure.

References