Modelling of flow in ejector

Martin Bílek[1]

[1] Vysoké učení technické v Brně, Fakulta strojního inženýrství, Energetický ústav, Technická 2896/2, Brno, maca_bilek@seznam.cz

Abstract This work deals with the modelling of flow in the ejector using the FLUENT software. It develops a thesis created in the past, where a mathematical flow model in the ejector was created and experimentally tested. The aim of this work is to analyze the ejector calculations and to model a flow in an ejector of the same shape and under the same conditions as in the experiment. Finally, the disagreement is discussed between the experimental figures and the ones gained from the mathematical model of the ejector.

1 Introduction

In the previous diploma thesis a graph of theoretical efficiency was made and the ejector devised [4]. This ejector was used in experimental measurements in a laboratory. On the basis of figures from the laboratory measurements a graph of efficiency was made. Both of these graphs should be the same or similar but they show quite big differences.

Therefore this work uses CFD methods to try to explain discrepancies between the mathematical model and the experiment [1].

2 Mathematical models

There were 2 mathematical models used for designing ejector and making the graphs of efficiencies.

One of these mathematical models of ejector took into account the efficiency expressed in formula (1) [3]. The graphs of efficiencies made from measured values in a laboratory and values gained from CFD modelling in the ejector were constructed according to this formula [3]. The quantities which are in formula (1) are shown in **Fig.1**.

$$\eta = \frac{Q \cdot \rho \left[\left(\frac{p_v}{\rho} + \frac{v_v^{\,2}}{2} \right) - \left(\frac{p_s}{\rho} + \frac{v_s^{\,2}}{2} \right) \right]}{Q_p \cdot \rho_p \left[\left(\frac{p_p}{\rho_p} + \frac{v_p^{\,2}}{2} \right) - \left(\frac{p_v}{\rho_v} + \frac{v_v^{\,2}}{2} \right) \right]} \tag{1}$$

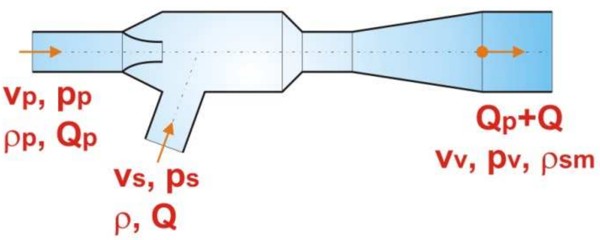

Fig. 1 Quantities from the formula (1)

The second mathematical model of ejector took into account the efficiency expressed in formula (2) [2]. And graph of efficiency was made on the basis of this formula which actually represents an efficiency of a draft tube. The quantities which are in formula (2) are shown in **Fig.2**.

$$\eta_s = \frac{\frac{p_k}{\rho} - \frac{p_3}{\rho}}{\frac{Q_3^{\ 2}}{2 \cdot S_k^{\ 2}} - \frac{Q_3^{\ 2}}{2 \cdot S_3^{\ 2}}} \tag{2}$$

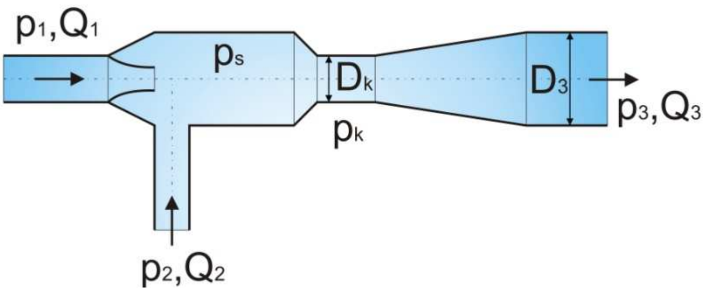

Fig. 2 Quantities from the formula (2)

3 Modelling of flow in the ejector

The flow in the ejector was solved with a one-phase 3D turbulent flow.

3.1 Geometry

The 3D model of flowing fluid in the ejector was created using SolidWorks software (**Fig.3**) according to previous plans of the ejector.

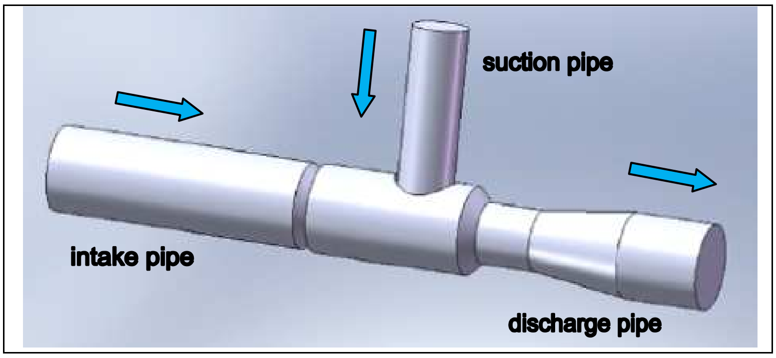

Fig. 3 Model of flowing fluid in the ejector with the directions of flow [1]

3.2 Boundary conditions

Boundary conditions which were used in the 3D model of the ejector are shown in **Fig.4**.

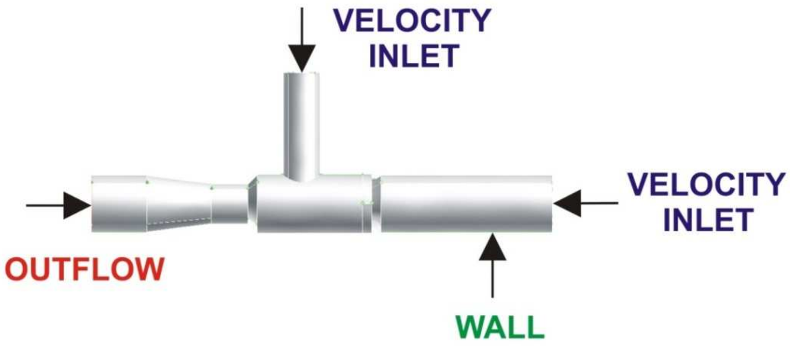

Fig. 4 Boundary conditions

3.3 Input values

Values from the experiment were used as input values for CFD computational problems, namely flow rates measured in the intake pipe and suction pipe of the ejector. Velocities were computed using flow rates which then defined boundary conditions of the VELOCITY INLET in FLUENT type.

4 The results of the CFD modelling

4.1 Graphs

The results of flow modelling in the ejector are shown in **Fig.5**. Here it is possible to compare three shapes of efficiency. The first one is the highest, it is theoretical efficiency. The second one is the lowest and it is efficiency made from measured values in a laboratory. The last one is the efficiency made on the basis of values gained from CFD modelling in the ejector. The shape of efficiency made from CFD values should be the same or similar to the experimental shape of efficiency. Nevertheless, the higher the flow rate ratio, the higher the differences between the efficiency curves.

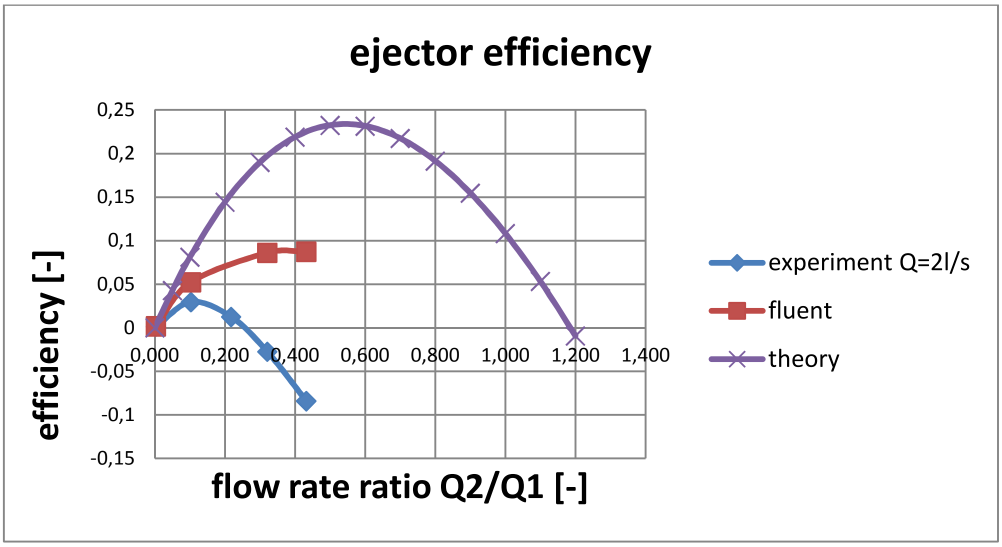

Fig. 5 Comparison shapes of efficiency

4.2 Graphic results

Fig.6 represents the flow in the ejector with a very low velocity in the suction pipe while **Fig.7** shows the flow in the ejector with much higher velocity in suction pipe. In some cases the flow with higher velocity in the suction pipe makes the stream of fluid which flows out of the nozzle bend. This bending of the stream fluid creates vortices in the diffuser.

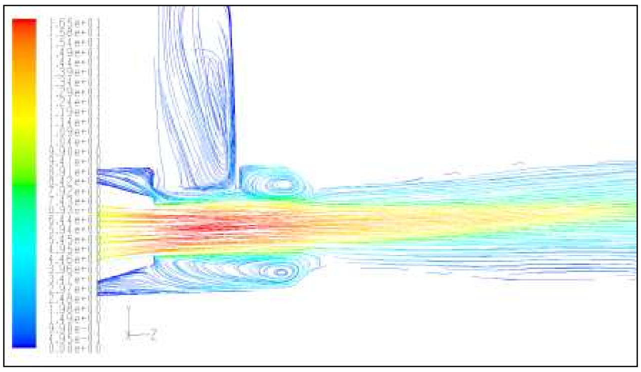

Fig. 6 Flow in ejector without vortices in diffuser [1]

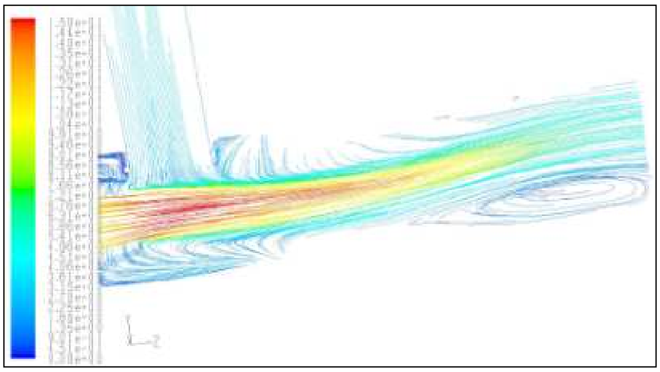

Fig. 7 Flow in ejector with vortices in diffuser [1]

5 Conclusions

The CFD flow modelling in the ejector has not confirmed the results from the experiment. Nevertheless they all have something in common. The higher the flow rate in the suction pipe, the bigger the differences from the theoretical efficiency. This fact can cause the stream of fluid out of the nozzle to bend and consequently to create vortices in the diffuser. Possible protection against creating vortices in the diffuser could be to change suction, e.g. to suck liquid from more places in the mixing chamber or extend the length between the suction pipe and the axis of the nozzle.

References

[1] BÍLEK, M.: *Modelování proudění v ejektoru*. Brno, 2009. Diplomová práce. Vysoké učení technické v Brně, Fakulta strojního inženýrství, Ústav fluidního inženýrství Viktora Kaplana. Vedoucí diplomové práce doc. Ing. Jaroslav Štigler, Ph.D.

[2] DANĚK, M., HALUZA, M., ZUBIK,P.: *Hydraulické charakteristiky řady přímých kuželových difuzorů s různým stupněm a rozložením vstupní rotace. Průběžná zpráva o výzkumu*, Vysoké Učení Technické v Brně, Fakulta stavební, Sdružené vodohospodářské laboratoře, Brno 1991.

[3] NECHLEBA, M., HUŠEK, J.: *Hydraulické stroje*; První vydání, Praha SNTL 1996.

[4] STRMISKA, M.: *Experimentální ověření ejektoru a vytvoření matematického modelu*. Brno, 2008. Diplomová práce. Vysoké učení technické v Brně, Fakulta strojního inženýrství, Ústav fluidního inženýrství Viktora Kaplana. Vedoucí diplomové práce Ing. Vladimír Habán, Ph.D.