Transactions of the VŠB – Technical University of Ostrava, Mechanical Series
No. 1, 2014, vol. LX
article No. 1979

Jaroslav KRUTIL[*], František POCHYLÝ[**], Simona FIALOVÁ[***]

CFD MODELING TWO-PHASE FLOW IN THE ROTATIONALLY SYMMETRIC BODIES

CFD MODELOVÁNÍ DVOUFÁZOVÉHO PROUDĚNÍ V ROTAČNĚ SYMETRICKÝCH TĚLESECH

Abstract

The work summarizes the basic findings which result from numerical modelling of flow at the mixture of air and water, with consideration of laminar and turbulent flow. The attention is focused on the development of the velocity profile of the liquid, depending on adhesion coefficient and the degree of hydrophobicity of the surface. We considered the geometry of a straight circular pipe arranged in vertical and horizontal position. The solution focuses on the finite element method and the tool utilized to evaluate the results was numerical program ANSYS Fluent.

Abstrakt

Práce sumarizuje základní poznatky, které vyplývají z numerického modelování proudění směsi vody a vzduchu s uvažováním laminárního a turbulentního proudění. Pozornost je zaměřena na vývoj rychlostního profilu kapaliny v závislosti na adhesním součiniteli, a stupni hydrofobie povrchu. Bylo uvažováno s geometrií přímého kruhového potrubí, které bylo uloženo ve vertikální a horizontální poloze. Řešení je zaměřeno na metodu konečných prvků a nástrojem využitým k zhodnocení výsledků je numerický program ANSYS Fluent.

Keywords

ANSYS Fluent, adhesion coefficient, two-phase flow, numerical modelling.

1 INTRODUCTION

At present, the hydraulic machines are required to work in a wide operating range. If the hydraulic machines operate outside their optimum operating point, local instabilities may occur. Such instabilities lead to the formation of vortical structures, which may cause expelling of a certain quantity of gas or, conversely, may cause the sucking in of water with a large amount of undissolved air.

2 MATERIALS AND METHODS

The ANSYS Fluent Software addresses the behaviour and properties of fluids using basic equations: the continuity equation and equations of motion. This numeric program is using several models that allow modelling a large number of separate but interrelated phases. These models are called multiphase models; the program offers the following three multiphase models [5], [7].

[*] Ing., Ph.D, Institute of Power Engineering, Faculty of Mechanical Engineering, Brno University of Technology, Technická 2896/2; 616 69, Brno; CZ, e-mail: jkrutil@seznam.cz
[**] prof., Ing., CSc, Institute of Power Engineering, Faculty of Mechanical Engineering, Brno University of Technology, Technická 2896/2; 616 69, Brno; CZ, e-mail: pochyly@fme.vutbr.cz
[***] Ing., Ph.D, Institute of Power Engineering, Faculty of Mechanical Engineering, Brno University of Technology, Technická 2896/2; 616 69, Brno; CZ, e-mail: fialova@fme.vutbr.cz

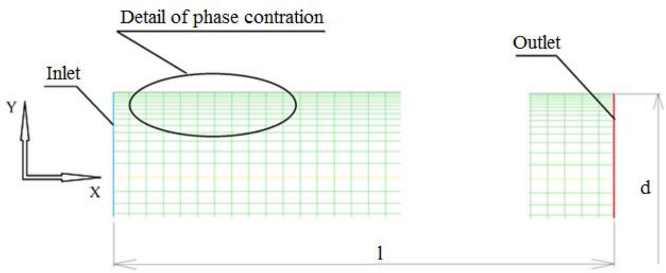
- VOF Model
- Mixture Model
- Euler´s Model

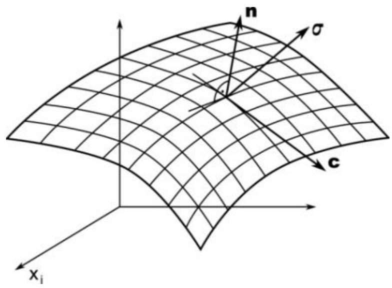
After a series of test tasks, it was found that the best model was the Euler´s Model. This model allows modelling of interactive multiple separate stages which are mutually contiguous. Because the volume of one phase cannot be filled by another, we introduced the concept of phase volume fraction $(\alpha_1, \alpha_2, \dots \alpha_n)$. The sum of the volume fractions is equal to 1. The flow equations are then modified with volume fraction of each phase. Due to the interaction, resistance forces and buoyancy forces were taken into account as well. You can find a detailed description of all multiphase models in the ANSYS Fluent Software Manuals [1], [2].

2.1 Geometry and boundary conditions

All CFD calculations were made on the 2D geometry, which represents a straight pipe. A physical pipeline model is shown in Figure 1 with the following boundary conditions:

- the inlet is defined as the velocity inlet ($v = 0{,}3\ m/s$ for laminar flow, $v = 1\ m/s$ for turbulent flow),
- the outlet is defined as pressure outlet (air outlet into the environment, p = 101325 Pa),
- and the wall is considered as a fixed (stationary).





$$\sigma_{Ai} = \Pi_{jk} n_k n_j n_i - \Pi_{ij} n_j = -k c_i$$

Fig. 1 Geometry and model of pipeline

Fig. 2 Define adhesion coefficient

Condition of wettability was considered in some variants - see the mathematical description in Figure 2. Wettability means a slippage characteristic of a liquid on surface of a subject, whereas the relative velocity of the fluid at the wall is not equal zero. In this case, the wettability is proportional to adhesion coefficient $k\ [Pa.s/m]$ [6], [9], [10]. Condition of wettability was implemented in the calculation, using a user-defined function (UDF). For our task we considered three basic variants:

- no slip
- k = 0,01
- k = 0,001.

Because the flow involves water and air mixture, it is also necessary to define the content of air, ranging between $0 - 3\%$. Pipe dimensions are: length $\mathrm{l} = 1000$ mm; the diameter is equal to $\mathrm{d} = 50$ mm. The mathematical model is formed by structured computing grid consisting of rectangular elements. The computing grid is more condensed in the area around the wall. More information can be found in the literature [8].

2.2 Result and discussions

Our interest was focussed on the monitoring of air behaviour inside the respective area, and on the influence of air on the velocity field development. In all figures the x-axis denotes the pipe diameter. Table 1 shows a list of solved variants.

Tab. 1 List of solve variants

Variant type	Designation
The pipeline model is arranged in horizontal position.	Variant 1
The pipeline model is arranged in vertical position.	Variant 2
The pipeline model is arranged in vertical position and one wall is wettable.	Variant 3

Variant 1 - The pipeline model is arranged in horizontal position.

Fig 3 shows that the velocity profile in the laminar flow is greatly influenced by the amount of air content in the liquid. This air tends to rise upwards towards the upper wall of the pipe. This fact is confirmed by progression of the volume fraction of air - see Figure 5. This trend is not as much pronounced in the turbulent flow (Fig. 4 and 6).

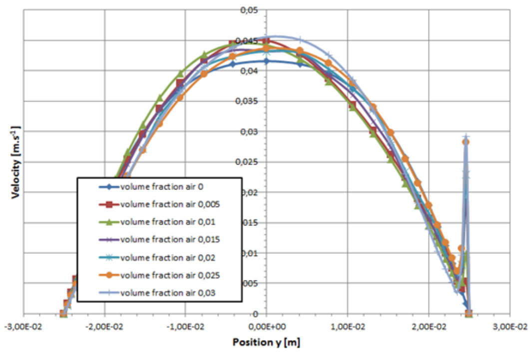


Fig. 3 Velocity Profile - laminar flow

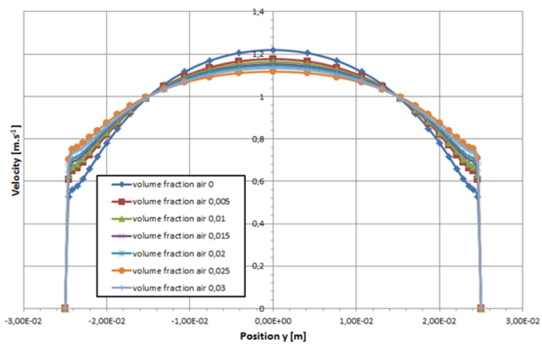


Fig. 4 Velocity Profile - turbulent flow

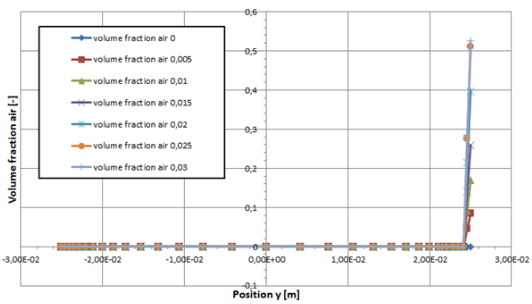


Fig. 5 Volume Fraction of Air - laminar flow

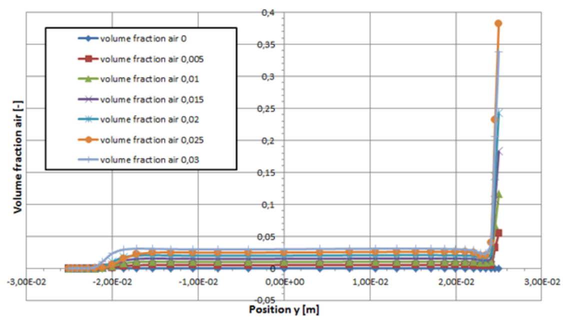
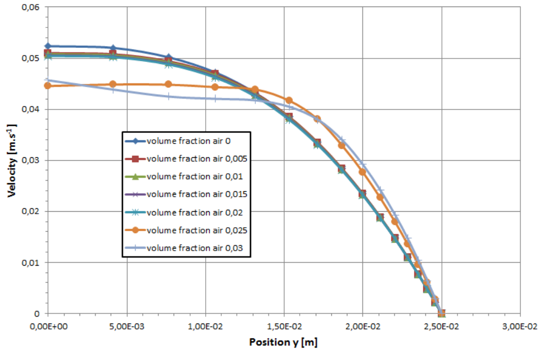


Fig. 6 Volume Fraction of Air - turbulent flow

Variant 2 - The pipeline model is arranged in vertical position.

This option considers upward fluid flow direction, which is opposite to the gravitational acceleration. For greater clarity, the profiles are shown only in one half of the pipe, since the results are axisymmetric. Velocity profiles (Figures 7 and 8) have a classical progression corresponding with the results reported in the professional literature [3], [4]. In this variant we can observe that the air content is almost equal to the set value at the input (i.e. from 0% to 3%) over the entire length of the pipe (see Figure 9 and Figure 10). Therefore its distribution is almost constant.



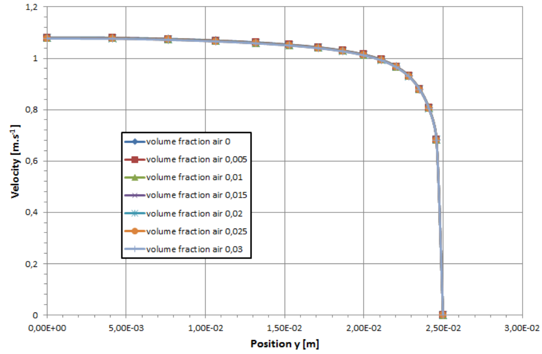
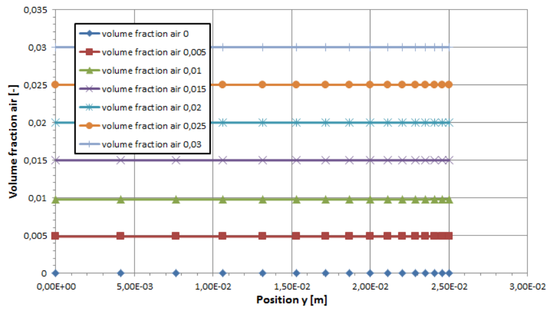


Fig. 7 Velocity Profile - laminar flow

Fig. 8 Velocity Profile - turbulent flow



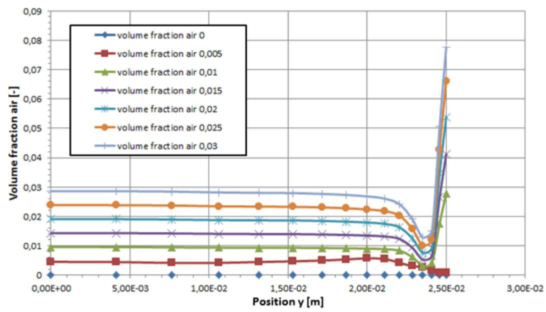
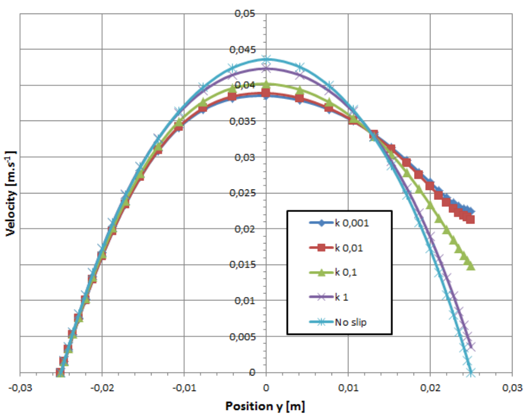


Fig. 9 Volume Fraction of Air - laminar flow

Fig. 10 Volume Fraction of Air - turbulent flow

Variant 3 - The pipeline model is arranged in vertical position and one side is wettable.

The results of the numerical simulation show how the k-coefficient affects the velocity profile of flowing liquid (water). It was found that the velocity profile is significantly deformed with the lower k-coefficient. This distortion is evident in the laminar and turbulent flow regimes - see Fig. 11 and Fig. 12. Comparison of velocity profiles is performed considering the volume 3% of air in water.



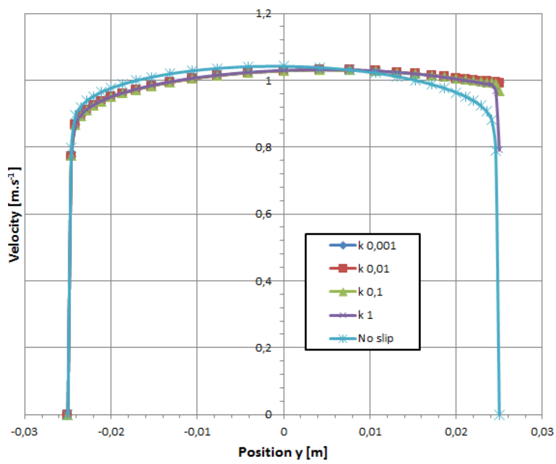
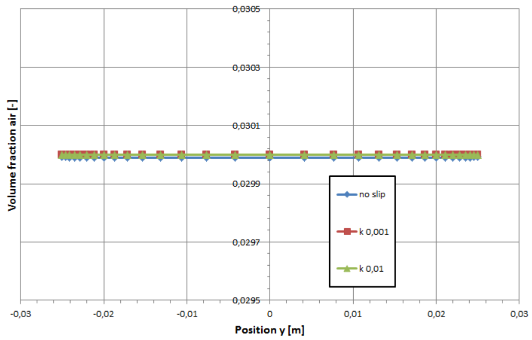


Fig. 11 Velocity Profile - laminar flow

Fig. 12 Velocity Profile - turbulent flow

Fig. 13 and 14 represent the comparison of air behaviour in the setting of wettability conditions (volume of air is 3%). It follows from these figures that air tends to increase its volume fraction at the higher value of the k-coefficient, and therefore adheres more to the wall.



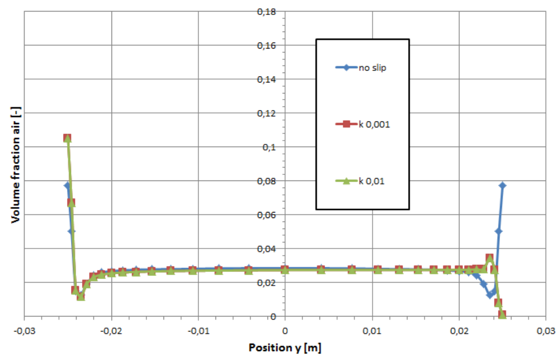


Fig. 13 Volume Fraction of Air - laminar flow **Fig. 14** Volume Fraction of Air - turbulent

The fact that the air tends to stick more to the wettable wall surface with the increasing *k*-coefficient is proved also by the following details - see Fig. 15 and Fig. 16. Location of each detail is shown in Fig. 1.

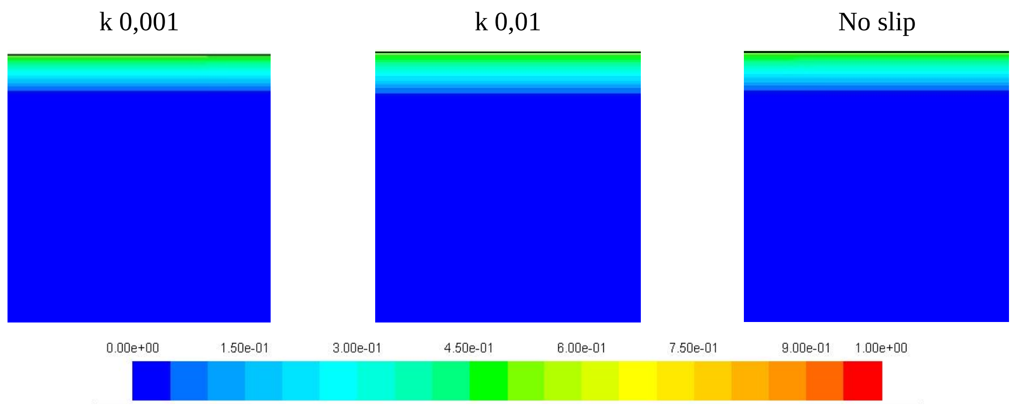


Fig. 15 The volume fraction of air in laminar flow, depending on adhesion coefficient k

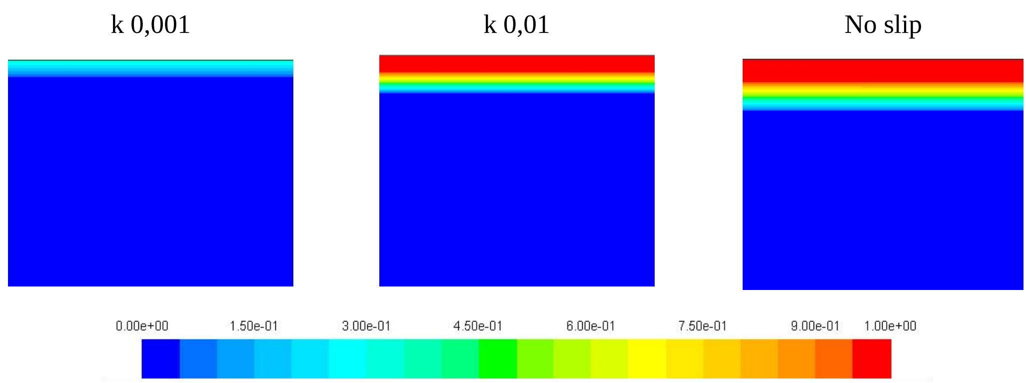


Fig. 16 The volume fraction of air in turbulent flow, depending on adhesion coefficient k

3 CONCLUSIONS

If we consider the two-phase flow, then the conservation law can derive significant evidence for non-stationary behaviour of the vapour phase ρ^{v}:

$$\frac{\partial \rho^{v}}{\partial t} \doteq grad\rho^{v} \cdot \boldsymbol{v}. \qquad (1)$$

The equation (1) shows that the stationary behaviour of the gaseous phase occurs only if $\mathrm{grad}\rho^{\mathrm{v}} = 0$, or if the density gradient is perpendicular to the vector of the resultant velocity $\boldsymbol{v}$. Considering this condition, we find that stable gas phase occurs only in the Variant 2; in all other cases it is unstable and time-dependent. A significant effect on the stability of the gaseous phase have spinning of the liquid, shape of the area, and direction of the flow related to gravity. The width of the gas layer and the stability are also greatly affected by adhesion coefficient k.

ACKNOWLEDGEMENT

The acknowledgement should be stated here, e.g. The work has been supported by the grant project: GA101/13-20031S, CZ.1.07/2.3.00/30.0005.

REFERENCES

[1] ANSYS FLUENT INC. Fluent 13.0, *Help –Theory guide*. Fluent Inc. 2010.

[2] ANSYS FLUENT INC. Fluent 13.0, *Help -User's guide*. Fluent Inc. 2010.

[3] BIRD, R. B. & STEWART, W. E. & LIGHTFOOT, N. N. *Transport Phenomena*. 2ed,Wiley, 2002, 914 s. ISBN 0-471-41077-2.

[4] BRENNEN, C. E. *Fundamentals of Multiphase Flow*. 1st Edition. USA: Cambridge University Press, 2005.345 p. ISBN-13978-0-521-84804-6.

[5] DOBEŠ, J. *Vliv teploty a tlaku čerpaného média na kavitační podmínky*. Diplomová práce. Ostrava: VŠB – Technická univerzita Ostrava, Fakulta strojní, Katedra hydromechaniky a hydraulických zařízení, 2012, 71 s. Přílohy: 73 s. Vedoucí práce: prof. RNDr. Milada Kozubková, CSc.

[6] FRODLOVÁ, B. & RUDOLF, P. & ZAVADIL, L. & KOZUBKOVÁ, M. & RAUTOVÁ, J. *Vliv částečné smáčivosti na proudění a kavitační oblast v Lavalově dýze*. In Sborník konference ANSYS 2011, 19. – 21.10.2011, Praha, 1. vyd. Praha: TechSoft Engineering, spol. sr.o.,2011. ISBN 978-80-905040-0-4.

[7] KOZUBKOVÁ, M. *Numerické modelování proudění FLUENT I*. [Online]. c2003. Ostrava: VŠB-TU Ostrava 116 s, poslední revize 6. 1. 2005, Dostupné z: <URL:http://www.338.vsb.cz/seznam.htm>.

[8] KRUTIL, J. & POCHYLÝ, F. & FIALOVÁ, S. *Two-phase flow in the rotationally symmetric bodies*. In Computational mechanics 2013, extended abstracts. Plzeň. 2013. p. 69 - 70. ISBN 978-80-261-0282-3.

[9] POCHYLÝ,F. & FIALOVÁ,S. & HABÁN,V. & RINKA, L. *The Wettability of the Liquid-Solid Interface*. In FIV2008 –FLOW- INDUCED VIBRATION, 9th International Conference on FIV, 30. 6. – 3. 7. 2008, Praha. ČR: Institute of Thermomechanics, AS CR, 2008, p. 47–52, ISBN 80-87012-12-7.

[10] POCHÝLÝ, F. & FIALOVÁ, S. & KOZUBKOVÁ, M. & ZAVADIL, L. *Assessment of cavitation creation depending on the surface wettability*. 25th IAHR Symposium on Hydraulic Machinery and Systems, 2010, vol. 12, no.1, p. 1-9.