Transactions of the VŠB – Technical University of Ostrava, Mechanical Series
No. 1, 2009, vol. LV
article No. 1671

Jana RAUTOVÁ[*], Milada KOZUBKOVÁ[**]

THE SIMULATION OF FLOW IN THE DIVERGENT NOZZLE
MODELOVÁNÍ PROUDĚNÍ V LAVALOVÉ DÝZE

Abstract

The divergent nozzle is an adjutage. An input tube section is tapering and an output tube section is broadening. It enables an expansion of an effluent gas on pressure lower than critical back pressure, and thereby the increase of the velocity (leaving) on speed greater than critical. The gas running through the expansion nozzle is izoentropic (the size of its entropia is almost invariable). Practical use of the flow in expansion nozzle is at solving flow in the adjutage.

Abstrakt

Lavalová dýza je výtoková trubice, jejíž vstupní část se zužuje a na ní navazuje část s rozšiřujícím se průřezem. To umožňuje expanzi vytékajícího plynu na tlak nižší než kritický protitlak (tlak nasycených par), a tím i vzrůst výtokové rychlosti na rychlost větší než kritickou. Plyn proudící Lavalovu dýzou je izoentropický - míra jeho entropie je téměř neměnná. Praktické využití proudění v Lavalově dýze je při řešení prodění v tryskách.

1 INTRODUCTION

In our case the water circulated by divergent nozzle. The pressure decreases, thanks to flow through the contraction part, on value saturated vapour pressure and that's why fluid flaskes it is cavitations. Owing to expansion the velocity decreases and on the other hand pressure increases, it means the primary situation [1], [3], [5].

The variants were solved [7], [9]:

A water and vapour,

B water, vapour and air.

2 EXPERIMENT

The experimental installation was created at the Energy Institute - Victor Kaplan Dept. of Fluid Engineering, Faculty of Mechanical Engineering, Brno University of Technology.

Pump, sensors of the pressure, flowmeter (orifice), divergent nozzle, tank and tube were used in circuit. Water was running medium. Detailed description of jet gives before simulation (see Fig. 1).

[*] Ing. Ph.D., Department of Hydromechanics and Hydraulic Equipment, Faculty of mechanical Engineering, VŠB-Technical University of Ostrava, tř. 17. listopadu 15, Ostrava, tel. (+420) 59 732 4385, e-mail jana.rautova@vsb.cz

[**] doc. RNDr. CSc., Department of Hydromechanics and Hydraulic Equipment, Faculty of mechanical Engineering, VŠB-Technical University of Ostrava, tř. 17. listopadu 15, Ostrava, tel. (+420) 59 732 3342, e-mail milada.kozubkova@vsb.cz

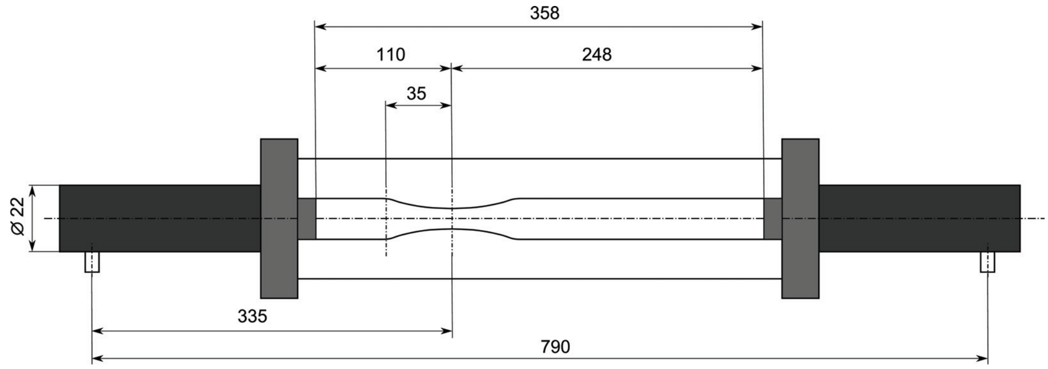


Fig. 1 Geometry of the area

Evaluation of experiment

The flow was regulated by the help of valve from closure to opening. The flow, absolute pressure before and behind divergent nozzle and atmospheric pressure were measured. The photos, describing rise and quantity vapour in divergent nozzle, were created during of the measuring. Variant of solving in this article is measuring number 18, where these values were found:

inlet pressure	$p_1 = 292849\,Pa$
outlet pressure	$p_2 = 115434\,Pa$
atmospheric pressure	$p_a = 98317\,Pa$
volumetric flow	$Q_v = 1{,}78\,l.s^{-1}$
quantity vapours	see Fig. 2

These values were used on numerical calculation for entry values.

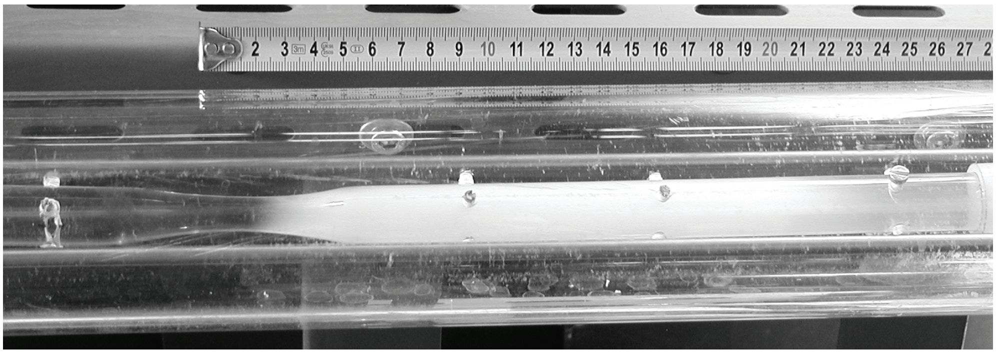


Fig. 2 Size cavitations areas in divergent nozzle (variant number 18)

3 MATHEMATICAL MODEL

Two-equation $k-\varepsilon$ model is recommended for calculation cavitations in literature [2], [4], [6] and [8]. The RNG $k-\varepsilon$ theory is advisable used for low-Reynolds number. This RNG $k-\varepsilon$ turbulence model is derived from Navier-Stokes equations. Equations are defined by mean value (pressure, velocity) and find them in literature [2], [4], [6] and [8].

The continuity equation applies to mean value:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \left(\rho \bar{u}_j\right)}{\partial x_j} = 0 \tag{1}$$

The equation for transmission momentum:

$$\frac{\partial\left(\rho\bar{u}_i\right)}{\partial t}+\frac{\partial\left(\rho\bar{u}_i\bar{u}_j\right)}{\partial x_j}=-\frac{\partial\bar{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\left(\mu+\mu_t\right)\frac{\partial\bar{u}_i}{\partial x_j}\right)+\underbrace{\rho\delta_{i3}g}_{\substack{\text{vztlakové}\\ \text{sily}}}+\underbrace{\rho f_c\varepsilon_{ij3}\bar{u}_j}_{\text{Coriolisov y sily}}+\rho f_i \tag{2}$$

Two-equation $k-\varepsilon$ model is complete equation for transmission turbulence kinetic energy k and velocity dissipation ε.

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial\left(\rho\bar{u}_j k\right)}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_k}.\frac{\partial k}{\partial x_j}\right)+\underbrace{\mu_t\left(\frac{\partial\bar{u}_j}{\partial x_l}+\frac{\partial\bar{u}_l}{\partial x_j}\right)\frac{\partial\bar{u}_l}{\partial x_j}}_{P}\underbrace{-g_j\frac{\mu_t}{\rho\sigma_h}\frac{\partial\rho}{\partial x_j}}_{G}-\rho\varepsilon \tag{3}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial\left(\rho\bar{u}_j\varepsilon\right)}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_\varepsilon}.\frac{\partial\varepsilon}{\partial x_j}\right)+\rho c_{1\varepsilon}\left(P+c_{3\varepsilon}G\right)-\rho c_{2\varepsilon}\frac{\varepsilon^2}{k} \tag{4}$$

where *P* and *G* are production turbulence kinetic energy due to tension and lift force.

Water and vapour eventually air make multiphase mixture. For simulation multiphase flow is possible use Mixture model. This model is advisable, when the velocity of individual phase translation is different. Model provide for solution changeover of phase, for the occasion are definition volumetric fraction of phase. Continuity equation for the mixture is:

$$\frac{\partial\rho_m}{\partial t}+\frac{\partial\left(\rho_m u_{m,j}\right)}{\partial x_j}=0 \tag{5}$$

where:

$u_{m,j}$ – is the mass-averaged velocity,

ρ_m – is the mixture density.

$$\rho_m=\sum_{k=1}^{n}\alpha_k\rho_k \tag{6}$$

where:

α_k – is the volume fraction of phase k,

n – is the number of phases.

The momentum equation for the mixture can be obtained by summing the individual momentum equations for all phases

$$\begin{aligned}\frac{\partial\left(\rho_m u_{m,i}\right)}{\partial t}+\frac{\partial}{\partial x_j}\left(\rho_m u_{m,i}u_{m,j}\right)=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu_m\left(\frac{\partial u_{m,i}}{\partial x_j}+\frac{\partial u_{m,j}}{\partial x_i}\right)-\mu\delta_{ij}\frac{2}{3}\frac{\partial u_{m,l}}{\partial x_l}\right)+\\ +\rho_m f_i+\frac{\partial}{\partial x_j}\left(\sum_{k=1}^{n}\alpha_k\rho_k u_{dr,km,i}u_{dr,,k,j}\right)\end{aligned} \tag{7}$$

where:

f_i – is the outer force by weight,

μ_m – is the viscosity of the mixture,

$$\mu_m = \sum_{k=1}^{n} \alpha_k \mu_k \tag{8}$$

$u_{dr,k,i}$ – is the slip velocity.

The grid for numerical solution

Quadrangle grid create, it has 9228 cell. I make subtilization on entry and grid has 9660 cell. The example is axially symmetrical.

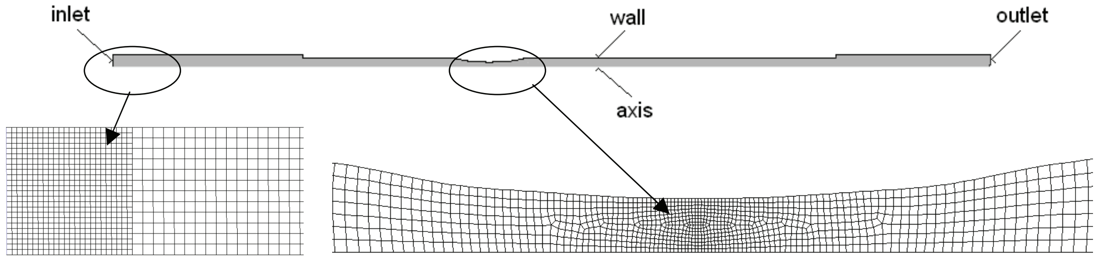


Fig. 3 The grid for numerical solution

Boundary conditions

mass-flow-inlet	$Q_m = Q_v \cdot \rho_{vody}$		
pressure-outlet	$p_2 = 115434\,Pa$		
saturated vapour pressure	$p_N = 3567{,}8\ Pa$		
surface tension coefficient	$\sigma\ = 0{,}717\,N \cdot m^{-1}$		
non-condensable gas mass friction	A	$f_A = 1{,}5 \cdot 10^{-8}$	(without air)
	B	$f_A = 1{,}5 \cdot 10^{-5}$	(with 2% of air)

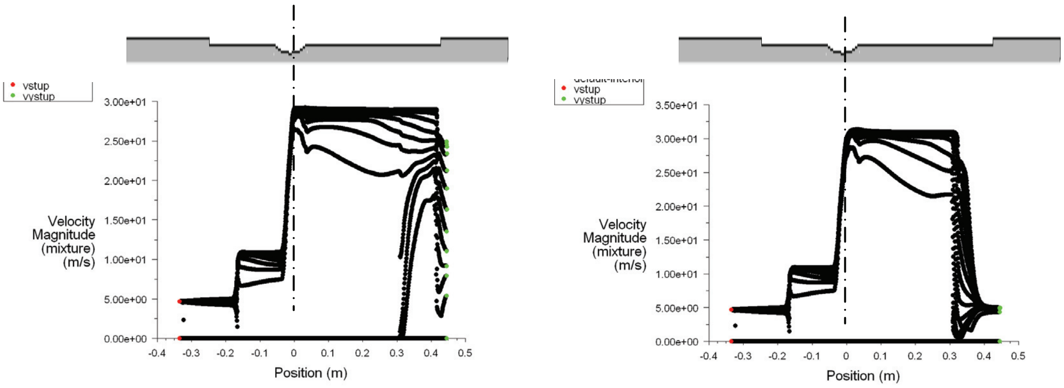
Physical properties

Temperature of water is constant and presuming its 20°C (i. e. 293,15K). Two variants are calculated because I don't know content air in water – without air and with 2% of air.

Tab. 1 Physical properties

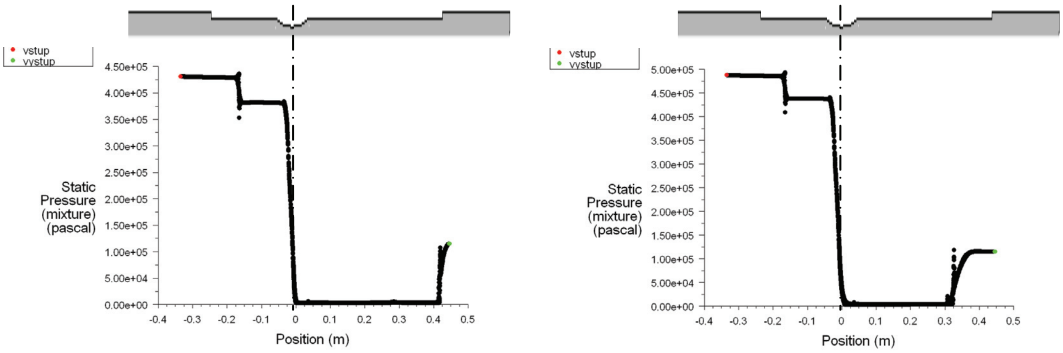
	density	viscosity
	$\left[kg \cdot m^{-3}\right]$	$\left[Pa \cdot s\right]$
water	998,2	0,001003
vapour	ideal-gas	kinetic-theory

Results of the numeric solution



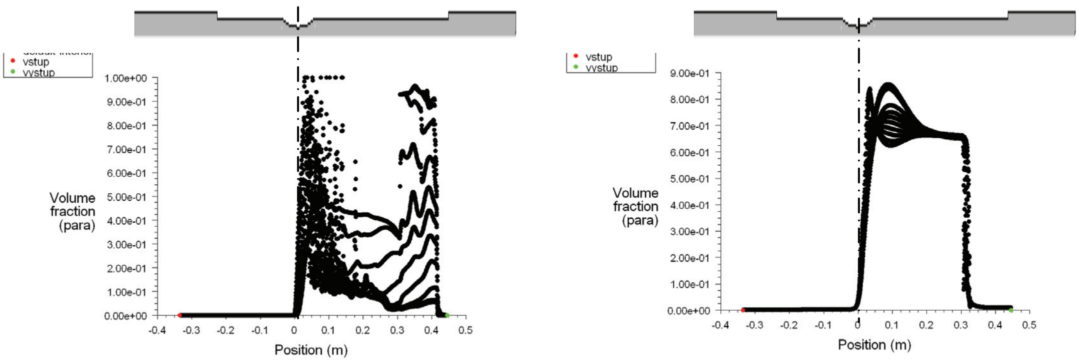
A B

Fig. 4 The development of the velocity



A B

Fig. 5 The development of the pressure



A B

Fig. 6 **Quantity of the vapour**

exp.

A

B

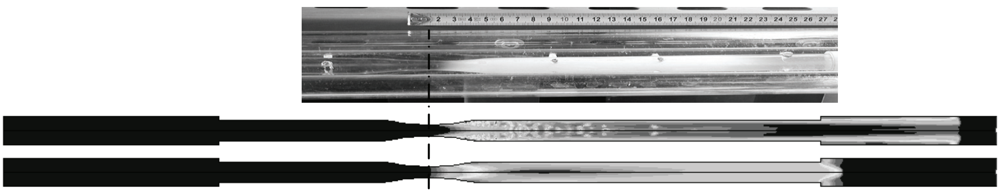


Fig. 7 Quantity of the vapour

Next variants for solving

Air (noncondensable gas) expressively influences flow. Therefore it will be necessary to test different quantity of the air in water and to verify the physical properties of both phases. Next values are defined in cavitations model - saturated vapour pressure and surface tension that they can influence the flow too.

Since Reynolds numbers are low, but mean tortuous flow was using turbulent model RNG $k-\varepsilon$ suitable for low Re numbers. Suitable laminar model would be advisable test.

At numerical solving were used like boundary conditions (entry - output) flow - pressure. At experiment was measured more value – pressure inlet and outlet, flow. Therefore it is possible use boundary conditions pressure – pressure, or less used pressure – flow.

4 CONCLUSIONS

Numerical solution of multiphase flow was solved by software Fluent 6.3.26

From results of numerical solving (see Fig. 4 - 7) is perceptible that it is preferable the solution B – i. e. the solution with 2% of air. Therefore it is necessary to be in more details engaged in physical properties of water and above all of air-volume in water.

From figure 4 – The development of the velocity – is discoverable that the exit speed at variants A isn't constant and is high. It follows the divergence of the mass flow rates on entry and exit, even if the volumetric fraction vapour on exit is zero, see Fig. 6 – on exit only water flows again.

From figure 5 - The development of the pressure – is perceptible that the pressure in part of a tube is equal to the pressure of a saturated vapour. It is typical for a vapour rise. From these running the section of the vapour rise is overvalue, therefore it is advisable to use Fig. 6 and 7 for a comparison the areas size.

From figure 7 - Quantity of the vapour – is perceptible that the section of the rise at variant A is very short and the vapour rises again behind the flange (therefore speed results are not acceptable at this variant) – which wasn't see at experimental measurement (exp.). Therefore the variant B is more suitable. There on the contrary the vapour section is overvalue (too long), which probably can influence with quantity of air in water. That's why it is necessary to engaged in physical properties of water and above all with air-volume in water.

GA ČR č. 101/09/1715 Kavitující vírové struktury vyvolané rotací kapaliny

REFERENCES

[1] BLEJCHAŘ, T. *Matematické modelování nestacionárního proudění, kavitace a akustických projevů v hydraulickém ventilu.* Dizertační práce. Ostrava:VŠB-TU Ostrava, 2005, 109 s.

[2] BOJKO, M. Charakteristika vícefázových matematických modelů při výrobě oceli v provozních kyslíkových konvertorech. In *Paper abstract of the XXVth International Conference of Departments of Fluid Mechanics and Thermomechanics Modra-Harmónia, Slovak Republic, 28 – 30 June, 2006*. Bratislava: Katedra tepelnej techniky, SjF STU v Bratislave, 2006, 9 s., ISBN 80-227-2434-3.

[3] BOJKO, M., KOZUBKOVÁ, M., RAUTOVÁ, J. *Základy hydromechaniky a zásobování hasivy.* SPBI Ostrava. 2007. 182 s. ISBN 80-86634-53-1.

[4] Fluent Inc. Fluent 6.3 – User's guide. [Online]. c2003.. Dostupné z: <URL:http://sp1.vsb.cz/DOC/Fluent_6.1/html/ug/ /main_pre.htm>.

[5] HRUŽÍK, L. Simulation of Pressure Response in Pipe. *Acta Hydraulica et Pneumatica,* 2007, (3), č.1, s. 38 – 42. ISSN 1336-7536.

[6] KOZUBKOVÁ, M. Numerické modelování proudění – FLUENT I. [Online]. c2003. Ostrava: VŠB – TUO, 116 s, poslední revize 6.1.2005, Dostupné z: <URL: http://www.338.vsb.cz/seznam.htm>.

[7] KOZUBKOVÁ, M., DRÁBKOVÁ, S. Modelování proudění oleje jako stlačitelné kapaliny (hydraulický ráz). In *12. uživatelská konference FLUENT 2006, 7. – 9. březen 2006.* Hrotovice: TechSoft Engineering, 2006, s. 153-160. ISBN 80-239-7211-1.

[8] KOZUBKOVÁ, M.: *Modelování proudění tekutin FLUENT, CFX*. Ostrava: VŠB-TU, 2008, 154 s., ISBN 978-80-248-1913-6, (Elektronická publikace na CD ROM)

[9] KOZUBKOVÁ, M.; ČARNOGURSKÁ, M. Physical and Mathematical Modelling of Liquid Flow Dynamics in Pipe (Hydraulic Shock). In *Sborník vědeckých prací VŠB-TU Ostrava, řada strojní, 2006, č.1,roč. LII.* Ostrava: VŠB-TU Ostrava, 2006, s. 103 – 114, ISBN 80-248-1142-1, ISSN 1210-0471.