Transactions of the VŠB – Technical University of Ostrava, Mechanical Series
No. 1, 2011, vol. LVII
article No. 1852

Milada KOZUBKOVÁ[*], Jana RAUTOVÁ[**]

MATHEMATICAL MODELLING OF FLUID FLOW IN CONE AND CAVITATION FORMATION

MODELOVÁNÍ PROUDĚNÍ V KUŽELU A VZNIK KAVITACE

Abstract

Problem of cavitation is the undesirable phenomena occuring in the fluid flow in many hydraulic application (pumps, turbines, valves, etc.). Therefore this is in the focus of interest using experimental and mathematical methods. Based on cavitation modelling in Laval nozzle results and experience [1], [2], [4], following problem described as the water flow at the outlet from turbine blade wheel was solved. Primarily the problem is simplified into modelling of water flow in cone. Profiles of axial, radial and tangential velocity are defined on inlet zone. The value of pressure is defined on the outlet. Boundary conditions were defined by main investigator of the grant project – Energy Institute, Victor Kaplan's Department of Fluid Engineering, Faculty of Mechanical Engineering, Brno University of Technology. The value of air volume was insignificant. Cavitation was solved by Singhal model of cavitation.

Abstrakt

V řadě hydraulických aplikací se vyskytuje problém kavitace jako nežádoucí jev, zvláště při proudění v čerpadlech, turbínách, ventilech atd. Proto je středem zájmu zkoumání jak experimentálními tak matematickými metodami. Na základě dosavadních výsledků a zkušeností při modelování kavitace v Lavalové dýze [1], [2], [4] byl specifikován problém proudění vody na výstupu z turbínového kola. Řešená úloha je zjednodušena na proudění vody v kuželu, kde na vstupu jsou definovány profily rychlosti v axiálním, radiálním a tangenciálním směru a na výstupu je zadán tlak. Okrajové podmínky jsou zadány řešitelem grantu - Energetický ústav, Odbor fluidního inženýrství Viktora Kaplana, Fakulta strojního inženýrství, Vysoké učení technické v Brně. Obsah vzduchu byl zanedbatelný. Pro řešení kavitace byl použitý Singhalův model.

1 MATHEMATICAL MODEL

Liquid can be exposed to pressure decreasing by constant temperature condition. The pressure may drop below the saturated vapour pressure, then the liquid column rupture is detected and the process is called cavitation. The liquid contains micro bubbles of non-condensable gas (dissolved), which under decreasing pressure may grow.

Mathematical model is defined for multiphase mixture consisted from water and vapour eventually air. For multiphase flow simulation the Mixture model is used. This model is advisable, when the velocity of individual phase translation differs. Model provides phase changeover, for this occasion the volumetric fractions of phases are defined.

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

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

Continuity equation for the mixture has the form:

$$\frac{\partial \rho_m}{\partial t}+\frac{\partial (\rho_m u_{m,j})}{\partial x_j}=0 \,, \tag{1}$$

where $u_{m,j}$ is the mass-averaged velocity and ρ_m is the mixture density defined by expression.

$$\rho_m=\sum_{k=1}^{n}\alpha_k \rho_k \,. \tag{2}$$

α_k is the volume fraction of phase k and n is the number of phases.

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

$$\frac{\partial (\rho_m u_{m,i})}{\partial t}+\frac{\partial}{\partial x_j}(\rho_m u_{m,i}u_{m,j})=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu_m\left(\frac{\partial u_{m,i}}{\partial x_j}+\frac{\partial u_{m,j}}{\partial x_i}\right)-\mu_m\delta_{ij}\frac{2}{3}\frac{\partial u_{m,l}}{\partial x_l}\right)+ \tag{3}$$

$$+\rho_m f_i+\frac{\partial}{\partial x_j}\left(\sum_{k=1}^{n}\alpha_k\rho_k u_{dr,km,i}u_{dr,,k,j}\right)$$

where f_i is the outer force by weight, $u_{dr,k,i}$ is used as the slip velocity and μ_m is the viscosity of the mixture:

$$\mu_m=\sum_{k=1}^{n}\alpha_k\mu_k \tag{4}$$

Two-equation turbulent $k-\varepsilon$ model is completed by equations for turbulence kinetic energy k and dissipation rate ε transfer:

$$\frac{\partial (\rho_m k)}{\partial t}+\frac{\partial (\rho_m u_{m,j}k)}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_k}.\frac{\partial k}{\partial x_j}\right)+\underbrace{\mu_t\left(\frac{\partial u_{m,j}}{\partial x_l}+\frac{\partial u_{m,l}}{\partial x_j}\right)\frac{\partial u_l}{\partial x_j}}_{P}\underbrace{-g_j\frac{\mu_t}{\rho_m\sigma_h}\frac{\partial \rho_m}{\partial x_j}}_{G}-\rho_m\varepsilon \tag{5}$$

$$\frac{\partial (\rho_m \varepsilon)}{\partial t}+\frac{\partial (\rho_m u_{m,j}\varepsilon)}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_\varepsilon}.\frac{\partial \varepsilon}{\partial x_j}\right)+\rho_m c_{1\varepsilon}(P+c_{3\varepsilon}G)-\rho_m c_{2\varepsilon}\frac{\varepsilon^2}{k} \tag{6}$$

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

The cavitation model is compatible with all turbulent models in Fluent.

The cavitation model can be used with multiphase Mixture model with or without slip velocity. The system includes two phases (water and vapour). The mass fraction of non-condensable gas is known beforehand and is assumed always as compressible.

The main liquid – water contains small amount of non-condensable gas (dissolved) [3]. This gas can has significant effect on the cavitation area due to expansion at low pressures. The model is assumed a mixture of the liquid (water) phase and gas phase. The gas phases consist of the water vapour and non-condensable gas, which was insignificant in our case.

The density of the mixture ρ is calculated as:

$$\rho=\alpha_v\rho_v+(1-\alpha_v)\rho_l \tag{7}$$

where ρ_l is the density of the liquid (water), ρ_v is the density of the vapour, $\alpha_l=1-\alpha_v$ is the volume fraction of the liquid (water) and α_v is the volume fraction of the vapour. The relationship between the mass fraction and volume fraction is:

$$\alpha_k=f_k\cdot\frac{\rho_m}{\rho_k} \tag{8}$$

where f_k is the mass fraction of the phase k, α_k is the volume fraction of the phase k, ρ_k is the density of the phase k.

The continuity equation in Singhal cavitation model has been used for two phases:

liquid phase:

$$\frac{\partial}{\partial t}\left[(1-\alpha)\rho_l\right]+\frac{\partial}{\partial x_i}\left[(1-\alpha)\rho_l u_i\right]=-R$$

vapour phase:

$$\frac{\partial}{\partial t}(\alpha\rho_{\mathrm{v}}) + \frac{\partial}{\partial x_i}(\alpha\rho_{\mathrm{l}} u_i) = R$$

mixture:

$$\frac{\partial}{\partial t}(\rho_m) + \frac{\partial}{\partial x_i}(\rho_m u_i) = 0$$

and

$$R = \frac{3\alpha}{R_{\mathrm{B}}} \cdot \frac{\rho_{\mathrm{v}}\rho_{\mathrm{l}}}{\rho} \cdot \sqrt{\frac{2}{3} \cdot \frac{(p_{\mathrm{B}} - p)}{\rho_{\mathrm{l}}}},$$

where R_B, p_B are the radius of bubbles and pressure respectively.

2 GEOMETRY AND GRID

Investigated problem is defined as three dimensional case. Hexahedral scheme of the grid was created in software Gambit and used for numerical simulation. The grid consists of 40095 cells (please see **Fig. 1** and **Fig. 2**). The problem was solved numerically in software Fluent 12.

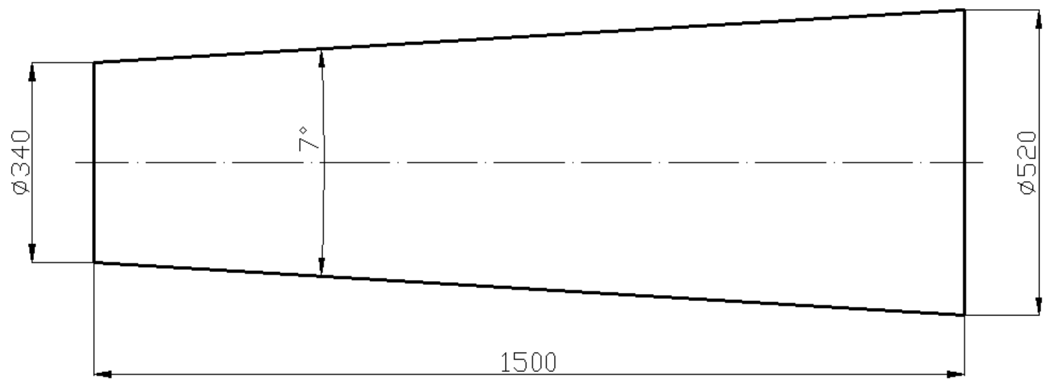


Fig. 1 Scheme of region geometry.

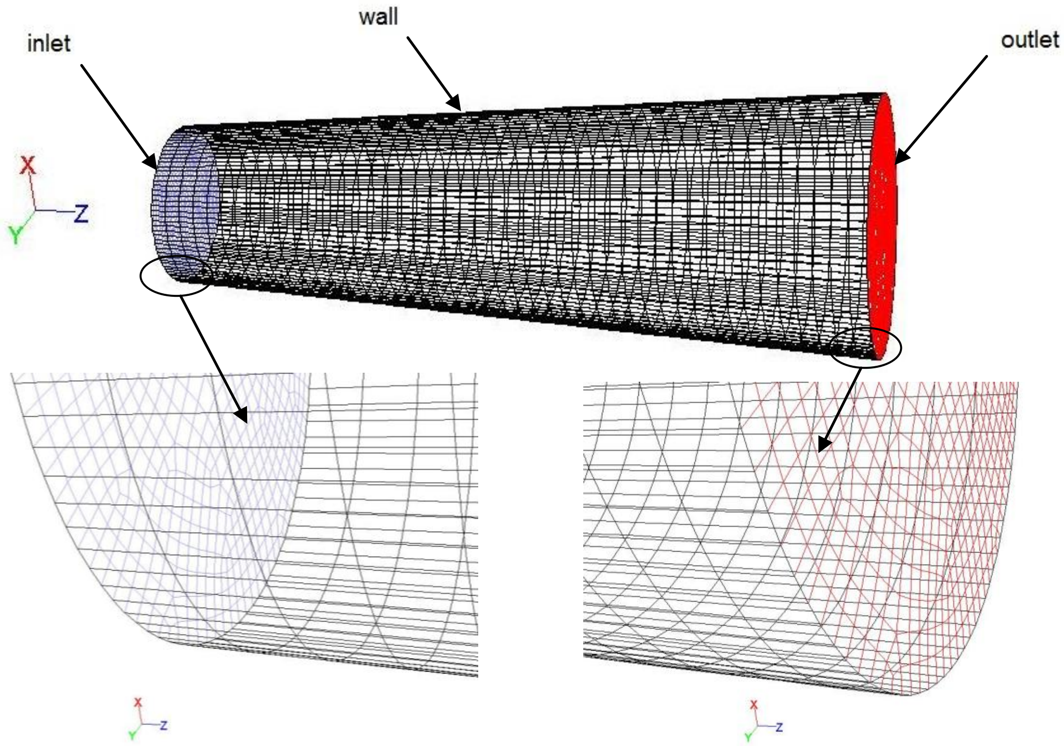


Fig. 2 The grid for numerical solution.

3 NUMERICAL PARAMETERS

3.1 Physical properties

Temperature of water is constant and equals 27°C (i.e. 300K), hence the physical model is supposed isothermal (recommended in literature). The physical properties are defined in Tab. 1.

Tab. 1 Physical properties

	density	viscosity
water	998.2 $kg \cdot m^{-3}$	0.001003 Pa·s
vapour	ideal-gas	$1.34.10^{-5}$ Pa·s

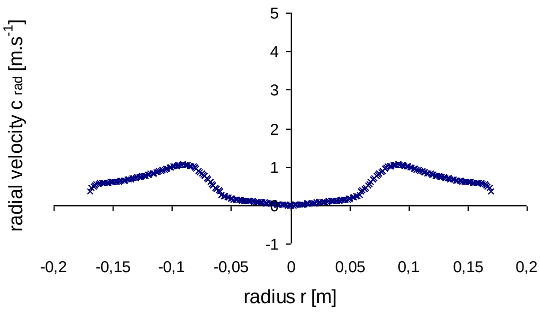
3.2 Boundary conditions

Boundary conditions on the inlet were defined in accordance with measured data (profiles of axial, radial and tangential velocities were added by VUT in Brno).

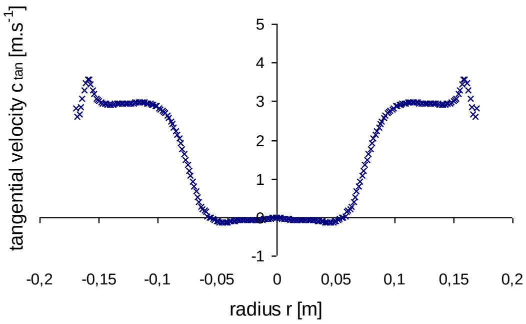
The problem is solved as multiphase flow, when water is the first phase and vapour is the second one. Due to the time dependent character of the vapour flow in cavitation region the solution is time dependent.

Tab. 2 Boundary conditions

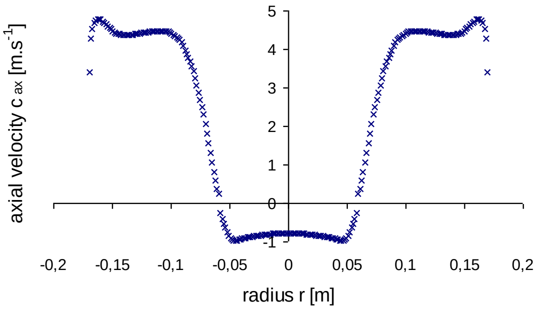
all regions	temperature		300 K
inlet - mixture	turbulent intensity		1%
	hydraulic diameter		0.345 m
inlet – water, vapour	velocity inlet	radial velocity	profile, Fig. 3 a)
		tangential velocity	profile, Fig. 3 b)
		axial velocity	profile, Fig. 3 c)
outlet - mixture	outlet pressure		10000 Pa
cavitation conditions	saturated vapour pressure		2367.8 Pa
	surface tension coefficient		0,0717 $N \cdot m^{-1}$
	non-condensable gas mass fraction		$1.5 \cdot 10^{-5}$



a)



b)



c)

Fig. 3 Velocity inlet.

a) radial velocity
b) tangential velocity
c) axial velocity

3.3 Numerical algorithm and discretization schema

Based on cavitation solutions in Laval nozzle experience the algorithms and discretization schema were chosen as mentioned in Tab. 3.

Tab. 3

numerical algorithm	discretization schemes		
	for pressure	for vapour	for other variables
Piso	Second Order	Quick	Second Order

It is necessary to solve the problem as unsteady state with time step of 0.001 s. The numerical solution was stopped at 95,161 iteration, when the mass flow rate on the inlet equals to mass flow rate on the outlet.

Tab. 4 Mass flow rate.

mass flow rate - inlet	329.4305 $kg \cdot s^{-1}$
mass flow rate - outlet	-335.0788 $kg \cdot s^{-1}$
difference	-5.6483 $kg \cdot s^{-1}$

Value of relative error is 1.715% (for the purpose of inlet and outlet mass flow rate evaluation). This value of error is insignificant.

4 RESULTS OF THE NUMERICAL SOLUTION

The numerical results are characterized by axial and tangential periodical flow of water. In the middle of the input part of the cone the cavitation rope is formed. It is rotated around the cone axes and moved in axial direction. This periodical character is observed in case of pressure, vapour and velocity field. It is obvious, that pressure minimum is given by saturated pressure. On the Fig. 4 the pressure distribution in three different times in axial direction and in three cross-sections supplemented by isosurface of saturated pressure is shown.

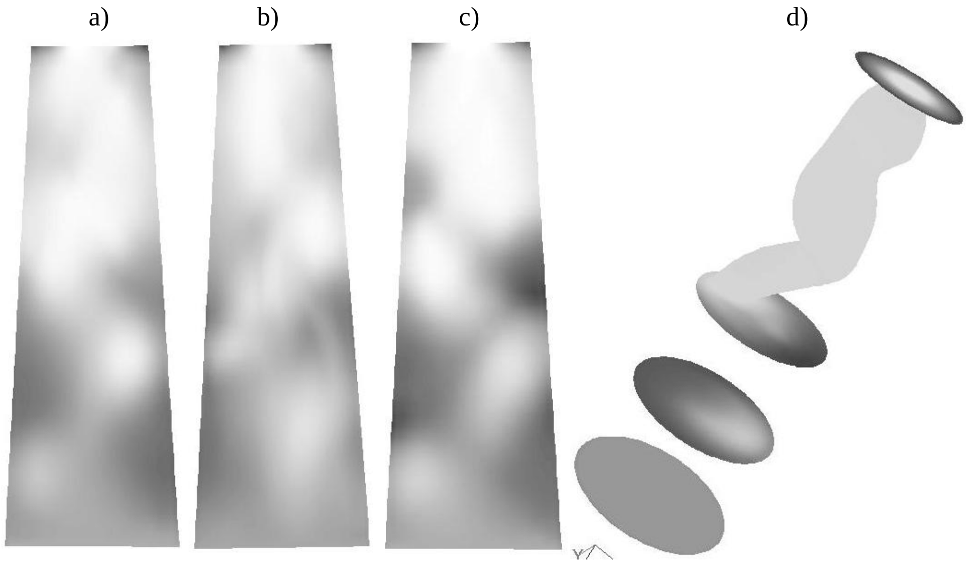


Fig. 4 Pressure distribution in section along the axes in different times (a), b), c) and in three cross-sections supplemented by iso-surface of saturated pressure (d)), $p \in \langle 2368, 16744 \rangle Pa$

In Fig. 5 in the same sections the value of vapour volume fraction is described.

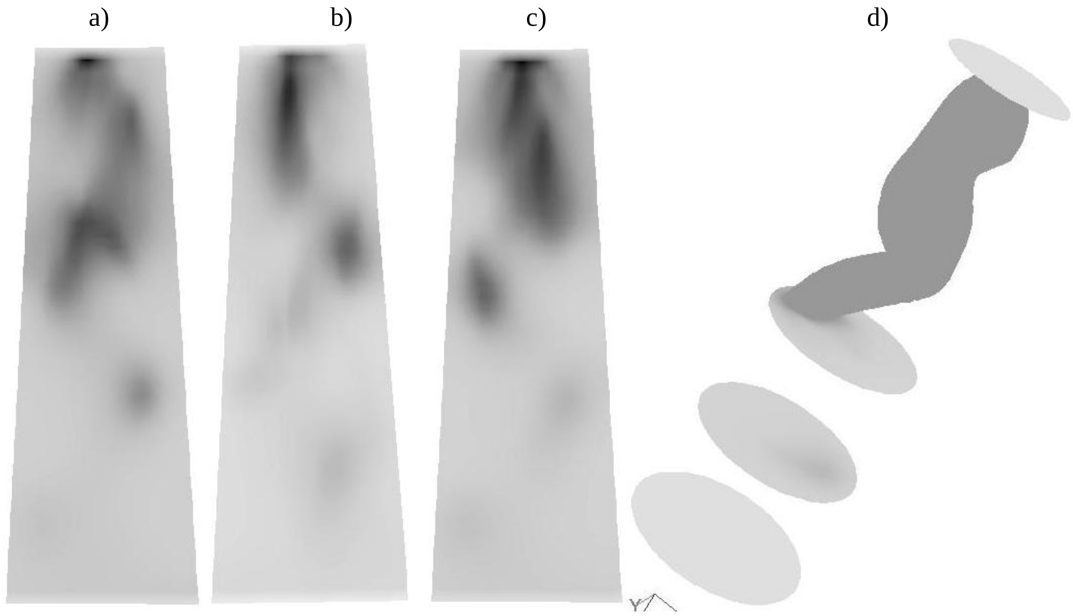


Fig. 5 Values of vapour volume fraction in section along the axes in different times (a), b), c) and in three cross-sections supplemented by iso-surface of volume fraction 0.5 (d)). $\alpha \in \langle 0,1 \rangle$

5 CONLUSION

The main goal of the work was to model mathematically the cavitating rope. Currently the problem was defined mainly as the water flow in cone to simulate simply the water flow on the turbine blade wheel outlet. Still this task was solved using one-phase model, where the existence of cavitation region was specified by pressure value lower then value of saturated pressure. This approach is illustrative because of non-realistic value of pressure and other hydraulic variables. Therefore the case was solved as multiphase flow. Profiles of axial, radial and tangential velocity were defined on the inlet, which were gained from experiment, see **Fig. 3**. Pressure condition at outlet was determined by numerical testing. Simulation results have shown that it is possible to reach the cavitation zone at pressure of 10 000 Pa on the outlet (please see **Fig. 4**). From evaluation of water vapour in section along the axes in different times (see Fig. 4 a), b), c)) and from space iso-surface (see Fig. 4 d)) it is evident, that the solution is time dependent and periodic. Similar behavior is noticeable on other hydraulic variables as pressure, velocity etc. By monitoring these value it was detected, that periodicity was observed in tangential as well as axial direction. It is interesting, that the solution was converged and the results respond the real idea using relatively small number of cells.

In the future the work will be extended to evaluation of pulsation flow and natural frequency definition. But it is computer time demanding problem. Numerical solution will be compared with physical experiment prepared by Energy Institute, Victor Kaplan‘s Department of Fluid Engineering, Faculty of Mechanical Engineering, Brno University of Technology.

GA ČR č. 101/09/1715 Cavitating Vortical Structures Induced by Rotation of Liquid

REFERENCES

[1] RAUTOVÁ, J.; KOZUBKOVÁ, M. Influence of air content in water on cavitation region in mathematical model. *Transactions of the VŠB – Technical University of Ostrava. Mechanical Series.* 2010, roč. LVI, č. 1, s. 291 - 300. ISSN 1210-0471.

[2] KOZUBKOVÁ, M.; RAUTOVÁ, J. Cavitation Modelling of the Flow in Laval Nozle. In *Proceedings of the 3rd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems*. 14 – 16. 10. 2009. Brno: Brno University of Technology, 2009, s. 583 – 592. ISBN 978-80-214-3947-4.

[3] 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>.

[4] RAUTOVÁ, J.; KOZUBKOVÁ, M. Matematické modelování proudění stlačitelné kapaliny s kavitací. In *Konference ANSYS 2009*. 23. – 25. 9. 2009. Plzeň: TechSoft Engineering, s.r.o., Praha, 2009, s. 163 – 172. ISBN 978-80-254-5437-4.

[5] 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>.

[6] 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.

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