OPTIMIZATION METHOD FOR DESIGNING OF HYDRAULIC ELEMENTS

MILADA KOZUBKOVA, JANA JABLONSKA, MARIAN BOJKO, PATRIK MARCALIK

Technical University of Ostrava, Departament of Hydromechanics and Hydraulic Equipment

Ostrava, Czech Republic

DOI : 10.17973/MMSJ.2018_10_201849

e-mail: milada.kozubkova@vsb.cz

Two methods are generally possible to design and optimize hydraulic components and devices.

The classic method is the experimental method. In the hydraulic laboratories, various models of components and devices are examined to understand their basic properties, to verify proposed assumptions, or to alter derived theoretical equations to equations that approximate reality, etc. In some cases, which are very difficult to solve theoretically, or even yet unsolvable, you can only get the values you need using an experiment. However, not all phenomena can be described through models.

Mathematical-physical modeling is a method by which Mathematical models based on the application of physical laws and phenomena can achieve the necessary results. These mathematical models consist of the definition of equations describing the given processes, which must be solved by means of numerical methods. Fluent, CFX Computerized software is used to solve the problem. Simulation can be performed within these softwares, which allows to evaluate different variables in a short period of time, to change the design of the element to suit the application, etc. However, it is a prerequisite to check the retained results by the experimental method.

The method of optimizing the parameters and shapes of products and equipment is already an integral part of the design process. This achieve product shape improvement without having to produce A number of Prototypes, you can create a variety of variants and perform simulations for different conditions. At present, the mathematical optimization method is based on the principle of adjunction, which is part of the ANSYS Fluent solution, which means saving time and finance while achieving qualitative improvement.

The article focuses on the theoretical and practical possibilities of using this method in the field of hydraulic elements.

KEYWORDS

optimization, goal function, CFD, Adjoint Solver

1 INTRODUCTION

The general fluid mechanics were historically divided into two areas, hydrostatics and hydrodynamics. Practical applications often involve both areas at the same time, [Incropera 2007].

The solution of flow problems can, in principle, be realized by mathematical and physical approach. But, both mathematically and physically, this is a complex problem. The solution of individual tasks is mostly separated, i.e. either by mathematical or physical approach. However, it seems very advantageous to deal with both approaches simultaneously and to combine the strong aspects of these approaches. The physical experiment provides basic information on the flowing fluid, geometry of the area, definition of the boundary conditions and the verification data. Verification data are possible at selected points in the area. The mathematical approach is extremely advantageous especially in terms of visualization of the area with the flowing fluid [Kozubkova 2009]. It allows inspection into the fluid together with the calculation of all important physical quantities in the whole area of the flowing field. This approach allows you to use simpler experimental measuring equipment even for complex tasks.

The mathematical model consists of the definition of transfer phenomena by means of the conservation laws of mass, momentum or other quantities. Because flow is a one-dimensional, planar, two-dimensional, axially symmetrical or generally three-dimensional and time-dependent, it is described by a system of integral or differential equations defined in the space, to be solved by numerical methods. Their use is conditioned by the need of extended knowledge of the field of flow, turbulence, numerical methods, computer technology, etc.

Modern engineering science can create and solve mathematical models of physical systems. Students often think skeptical about fluid mechanics, as the basic mathematical models seem very complicated and difficult for applications. But for example, a simplified hydrostatic theory can be used to determine the distribution of pressure in the area. It is true that the fluid dynamics described by Navier - Stokes equations are complex and simple solution, except for minor excercises, doesn´t exist.

The last period is characterized by the rapid development of numerical methods for flow modelling, both laminar and turbulent. In this context, several quality software products are created that can be used. Part of them is dedicated to the one-dimensional flow of liquids and is mainly usable in the solutions of fluid systems, which consist of fluid elements and piping system [Kozubkova 2009a]. The mathematical model is based on the electro-hydraulic analogy (Flowmaster, the extension of Matlab simulink, ie. SimHydraulics, AMESIM).

Other software specialize in spatial flow and their use primarily falls into the area of construction and currently, even in mathematical optimization of fluid elements (ANSYS Fluent, CFX, STAR CC +). The result is the distribution of pressure and the velocity or flow of the entire solved area [Kozubkova 2009b], [Navratil 2016], [Soltys 2017]. The boundary conditions that significantly affect the solution must be considered.

2 THEORY OF OPTIMIZATION USING GRADIENT METHOD

The idea of Adjoint method is appearing everywhere in modern and classical mathematics. It dates back to the 18th century. It has only recently proved to be a strong means to extend engineering analysis using CFD methods. Adjoint Solver provides specific information about the fluid system, which is very difficult to obtain in another way, [Ansys 2013], [Ansys 2015]. It can be used to calculate derivations of the engineering quantities with respect to all inputs into the system. An example is

- Derivation of resistance with respect to the shape of the vehicle.
- Derivation of the total pressure gradient with respect to the shape of the flow path.

Adjoint Solver is a specialized tool that extends the analysis built into the standard (conventional) solver and provides detailed information on the sensitivity of the fluid system. In order to perform the simulation using the standard flow solvers of ANSYS, the user creates the geometry with the computational mesh, determines the properties of materials and physical models, configures the boundary conditions of different types. As soon as the standard solver converges, it provides a detailed set of data describing the state of flow. If a change is made to any of the data that define the problem, then the results of the calculation may change. The changes depend on how sensitive the flow is to the edited parameter. It is a specialized tool that extends the standard analysis, capable of providing a record of sensitivity of the system, that can be used to optimize the design of a given element. In fact, the derivation of the solution result according to this parameter is quantified by the sensitivity of the first order. Determination of derivatives is the essence of sensitivity analysis.

There are many optimization methods which suitability for each case is defined by the time demands of the calculation and the efficiency of handling many constructional variables. The gradient method is the best-known method that is capable of working with a large number of construction variables.

2.1 Methodology of solving Adjoint solver

As mentioned above, Adjoint Solver can be used only when a computational mesh is created, physical models, boundary conditions, etc. are set and data set, generated by a standard flow calculation, is available. The following resolution procedure is divided into several phases.

- Adjoint Solver Settings – it concerns with defining monitored variables, solution controls, and so on.
- Calculation of the sensitivity of the system against the specified variables - after completion of the calculations (if convergence occurs) there is a sensitivity data set which can be used to define the design change of the system.
- Morphing-Adjoint Solver - it allows you to precisely and easily determine which part of the geometry is to be adjusted. After you modify a shape, you do not have to create a new mesh, because the computational mesh automatically reshapes when geometry changes
- Standard Calculation – analysis of the flow in the new geometry.
- Calculation Repetition - if the calculation converges, it is possible to repeat the whole procedure as long as the residuals of both calculation of the sensitivity of the system and the standard calculation converges or until the change in design and the value of the monitored variable is found sufficient.

On Fig. 1 the solution methodology of the Adjoint Solver module is displayed [Tzanakis 2014].

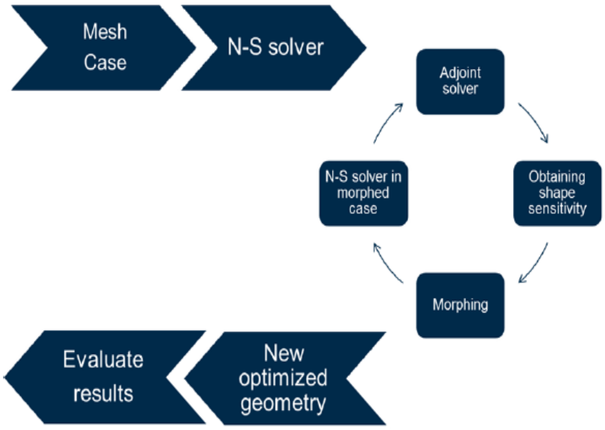


Figure 1. Methodology of the Adjoint Solver solution [Tzanakis 2014]

2.2 Restrictions on the use of Adjoint Solver

Adjoint Solver is a method that has a specific limitation and is based upon the following basis [ANSYS 2013]:

- The flow state is defined as a permanently incompressible single-phase flow that is either laminar or turbulent and lies in an inertial reference system.
- The basic flow must be solved for such boundary conditions, so that the task converges well and quickly (i.e. not too severely turbulent, so that vortex path doesn´t appear due to wrapping of obstacles to ensure sufficient pressure in the area of interest, because then Adjoint Solver will converge well).
- For turbulent flow the assumption of frozen turbulence is used, in which the effect of changes in the state of turbulence is not considered in the calculation of sensitivity.
- In turbulent flow, standard wall functions are used on all walls.
- Adjoint Solver uses methods that are first order of precision in space by default. Second-order precision methods can be selected.
- Boundary conditions are only the following type: wall, input velocity, output pressure, symmetry, rotational and translational periodic conditions

It is important to note that these requirements are not a restriction for basic flow solver, but they are a limitation for Adjoint Solver. For hydraulic and pneumatic tasks (i.e., the flow in closed areas) is advisable to use parts of solver relating to optimization of pressure gradient, while resistance and pressure forces are evaluated for wrapping tasks. Also, the combination of monitored parameters is very illustrative.

Stability problems can occur when applying an adjoint solution to tasks with a very fine computational mesh, complex geometry, or possibly on tasks with a high Reynolds number. These instabilities may be based on the irregularity of small dimensions in the fluid field or, possibly because of severe shear stress and tend to be limited to small and isolated parts of the flow area. If they are neglected, their presence may disrupt the entire Adjoint calculation despite the fact that the occurrence can occur only in several cells of the computational mesh. To obtain a adjoint solution in such cases, it is necessary to apply a stabilization scheme, [Navrátil 2016], [Šoltys 2017].

3. APPLICATION OF ADJOINT SOLVER WHEN OPTIMIZING GEOMETRY OF AN ELBOW

The task is to use the Adjoint Solver tool, which is a part of the ANSYS Fluent 18.2, to reduce the pressure loss and to smooth the velocity profile, i.e. to minimize the mean variance of the total pressure in the elbow (see Fig. 1).

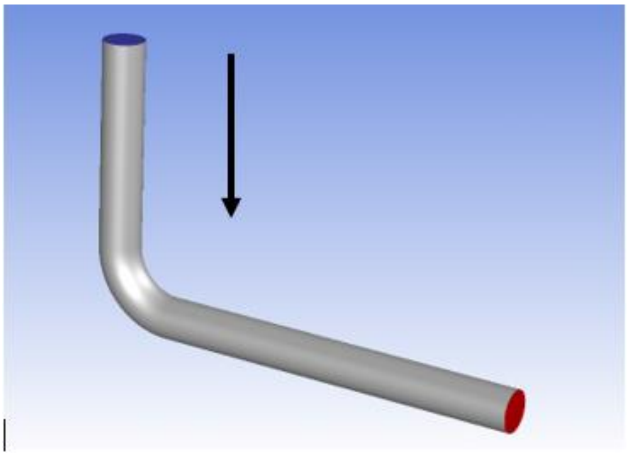


Figure 2. Geometry of elbow geometry

Area dimensions are shown in Tab. 1.

Dimensions	Units	Values
Input diameter d_1	m	0.1
Output diameter d_2	m	0,1
Input length l_1	m	1,1
Output length l_2	m	0,7
Elbow radius R	m	0,2

Table 1. Area dimensions

The flowing medium is air with constant physical properties (relative to flow velocity and Mach number). i.e. density ρ is 1,225 kg. m^{-3} and dynamic viscosity μ is 1,7894.10^{-5} Pa.s.

Boundary conditions are specified at input (Tab. 2) and output (Table 3). Wall conditions are defined as no-slip conditions.

Input (Velocity inlet)	Units	Values
input velocity u	m.s^{-1}	10
intensity of turbulence I	%	2
hydraulic diameter d_h	m	0,1

Table 2. Input boundary conditions

Output (Pressure outlet)	Units	Values
static pressure p	Pa	0
intensity of turbulence I	%	2
hydraulic diameter d_h	m	0,1

Table 3. Output boundary conditions

The flow is assumed to be stationary, isothermal, turbulent (k-ε standard model, standard wall function).

The first step is calculation of the fluid field, while the task must converge as stationary, i.e. all resolved variable residuals must be less than 0,001 (see fig. 3.)

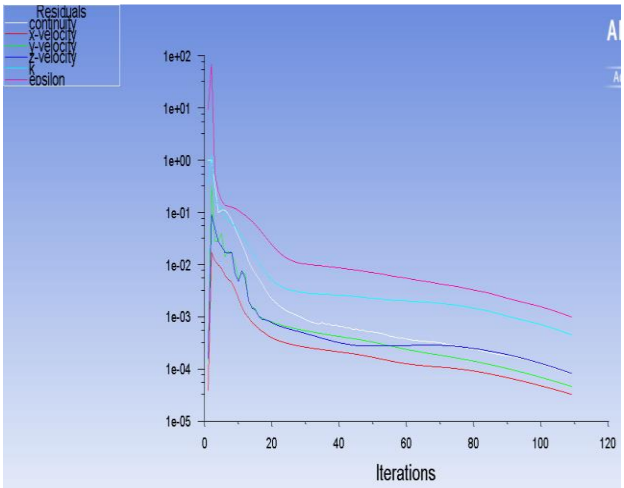


Figure 3. Residuals of basic solution of fluid field

The pressure loss at steady air flow was evaluated as a difference of the mean value of the static pressure on the inlet and outlet **Δp = 32.5 Pa**. With regard to the minimizing this pressure loss, optimization will take place.

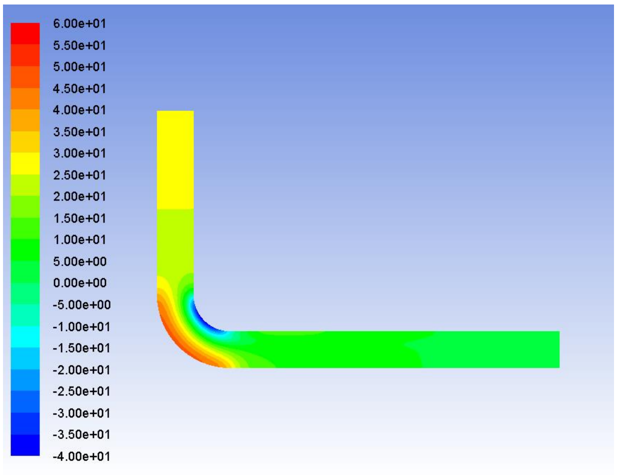


Figure 4. Distribution of static pressure in axial cross-section [Pa]

By setting the Adjoint Solver parameters, appropriate methods of solution and, where appropriate, stabilization methods, it should end in a convergent solution. The sensitivity map is the prediction of geometry optimization and it shows appropriate places where it has sense to make changes to the geometry due to the minimization of goal function, which is defined as:

$$f = |p_{inlet} - p_{outlet}| + var\sqrt{p_{total}}$$

see Fig. 5.

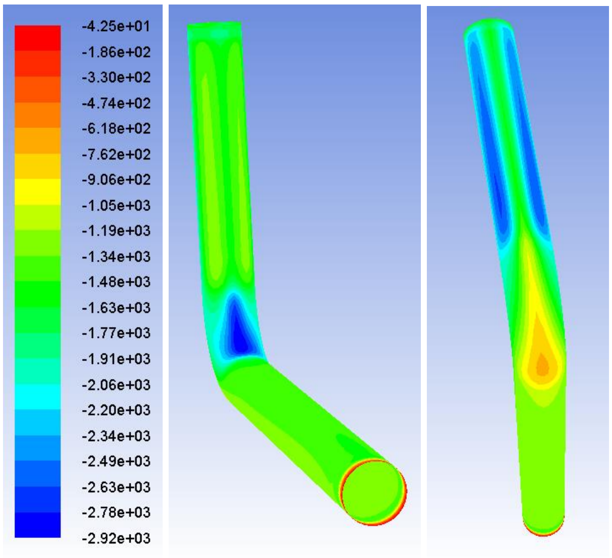


Figure 5. Maps of sensitivity on the wall of the pipe

From maximal values of sensitivity follows the definition of the deformation area (see fig. 6).

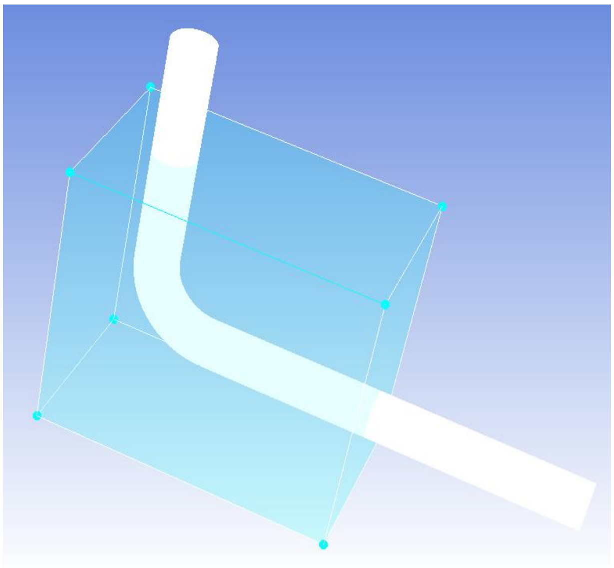


Figure 6. Specification of deformation area

Already at this stage you can determine the expected change of goal function. This value is estimated and only refined after the optimization by calculating the full fluid field. If this value is sufficient for the user, the geometry is changed, and a preview of the modified geometry is displayed. Deformation can be repeated, while monitoring the convergence of Adjoint Solver, the value of the expected pressure gradient and the reality of the new geometry, see Fig. 7.

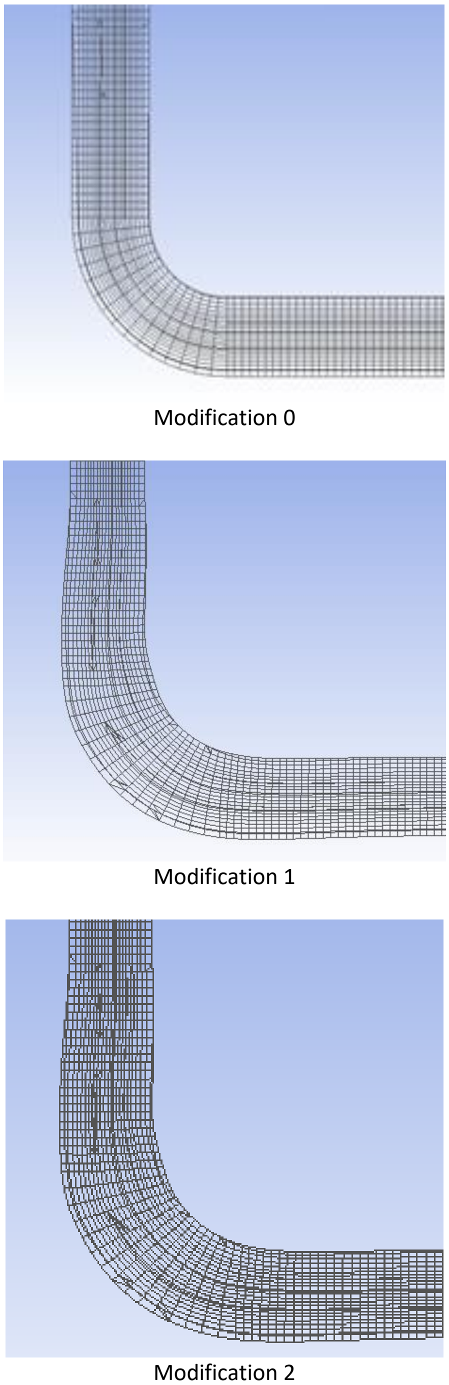


Figure 7. Modification 0 - original geometry, modification 1 - geometry deformation after first calculation, modification3 - geometry deformation after second calculation

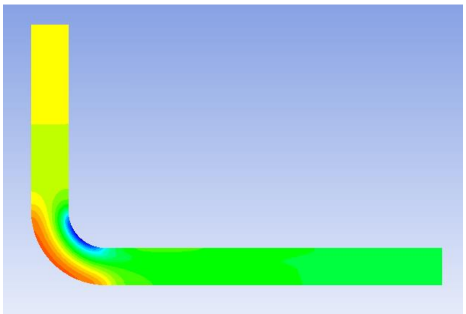
The effect of the shape change on the pressure gradient and velocity distribution is checked by calculation of the basic fluid field using the original Navier -Stokes equations and the results of pressure are displayed on Fig. 8.

In the table 4 there is the evaluation of changes of monitored pressure drop in relation to the default state and after the two iteration loops.

Modification	0	1	2
Δp [Pa]	28,3	24,5	21,7
Variance [%]	0	13,43	23,32

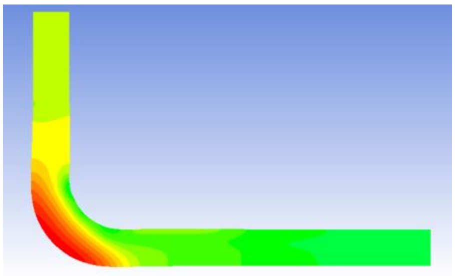
Table 4. Overview of the changes in pressure drop [Pa] and evaluation of variance according to original task [%]

In the last Fig. 9 the contours of velocity magnitude for three modifications are shown.

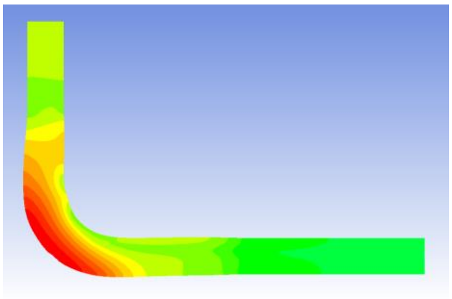


Modification 0

(original geometry with high negative pressure)



Modification 1



Modification 2

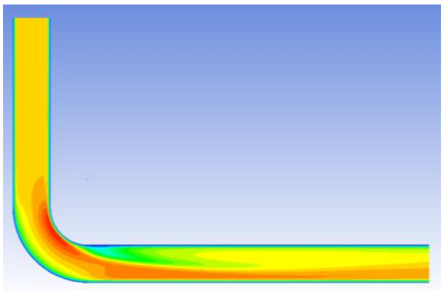
Figure 8. Distribution of pressure in axial cross-section [Pa]

4. CONCLUSION

Optimization of the geometry during fluid flow was so far solved by the method of trial and error, when, based on experience, a number of variants of the given geometry were created and subsequently tested. Optimization is currently a new trend in design of hydraulic elements. It uses numerical methods for spatial modelling of flow and numerical optimization methods. The paper focuses on a brief explanation of the principle and procedure of using the optimization method. It should be noted that the optimization should be carried out carefully so that the optimized element should have "reasonable shape", required properties and should be manufacturable. The consequent problem is to export of optimized geometry to the common CAD format.

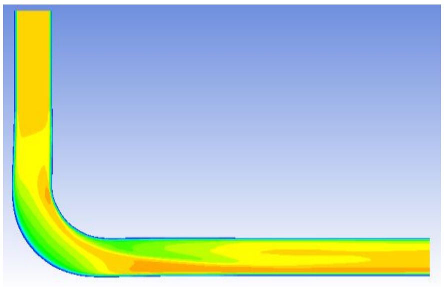
The condition of a stationary flow in a given geometry may appear to be a significant limitation of usability. It is then on the user's experience if he can solve the optimization only on selected parts of the element where this condition can be accepted, and after optimizing them join these newly modified parts together. Often a stationary solution predicts cavitation on the walls of the hydraulic elements. Even here, optimization can be an effective means.

The most common use of optimization is in the automotive industry in the applications of the flowing through various piping systems.

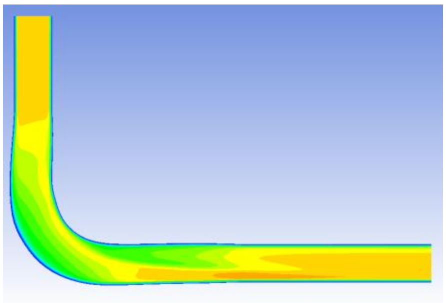


Modification 0

(original geometry with high velocity magnitude)



Modification 1



Modification 2

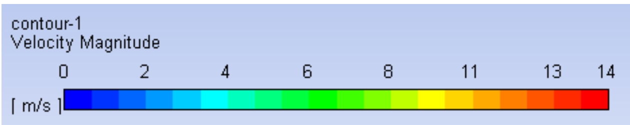


Figure 9. Distribution of velocity magnitude in axial cross-section [m.s^{-1}]

ACKNOWLEDGMENTS

The work presented in this paper was supported by a grant SGS „Modeling and Experimental Verification of Dynamic Phenomena in Fluid and Vacuum Systems" SP2018/57.

REFERENCES

[Ansys 2013] Ansys Fluent Manual. Ansys Fluent Adjoint Solver. Version 15.0. ANSYS, Inc., 2013. 86 s. [cit. 2017-05-07]

[Ansys 2015] Ansys Fluent Manual. Ansys Fluent Advanced Add-On Modules Version 16.2. ANSYS, Inc., 2015. 474 s. [cit. 2017-05-07]

[Incropera 2007] Incropera, F. et al. Fundamentals of Heat and Mass Transfer, 6. edition, John Wiley and Sons 2007, 996p., ISBN 978-0-471-45728-2

[Kozubkova 2009a] Kozubkova, M. Mathematical models of cavitation and hydraulic hammer (in Czech). Ostrava, VSB - TU Ostrava, 2009, pp 130, ISBN 978-80-248-2043-9.

[Kozubkova 2009b] Kozubkova, M. et al. Adjoint Solver applications on examples. Ostrava, 2015. Learning text. VŠB-TUO.

[Navratil 2016] Navratil, J. Numerical modelling with subsequent optimization of the shape to minimize pressure loss using adjoint solver. Ostrava, 2016. Diploma thesis. VSB-TUO. 80 p. Head of thesis Doc. Ing. Marian Bojko, Ph.D. Available from: http://hdl.handle.net/10084/115024.

[Soltys 2017] Soltys, D. Optimisation of flow parts of L10 valves. Ostrava, 2017. Diploma thesis. VSB-TUO. 71 p. Head of the thesis of Prof. RNDr. Milada Kozubkova, CSc. Available from: http://hdl.handle.net/10084/117493

[Tzanakis 2014] Tzanakis, A. Duct optimization using CFD software ' ANSYS Fluent Adjoint Solver '. Gothenburg, Sweden, 2014. Diploma thesis. Chalmers University of Technology in Gothenburg. 42 p. Available from: http://publications.lib.chalmers.se/records/fulltext/202020/202020.pdf

CONTACTS:

Prof. RNDr. Milada Kozubkova, CSc.
VSB Technical University of Ostrava
Departament of Hydromechanics and Hydraulic Equipment
17. listopadu 15/2172, 70830 Ostrava, Czech Republic
Phone: +420 597 323 342
e-mail: milada.kozubkova@vsb.cz