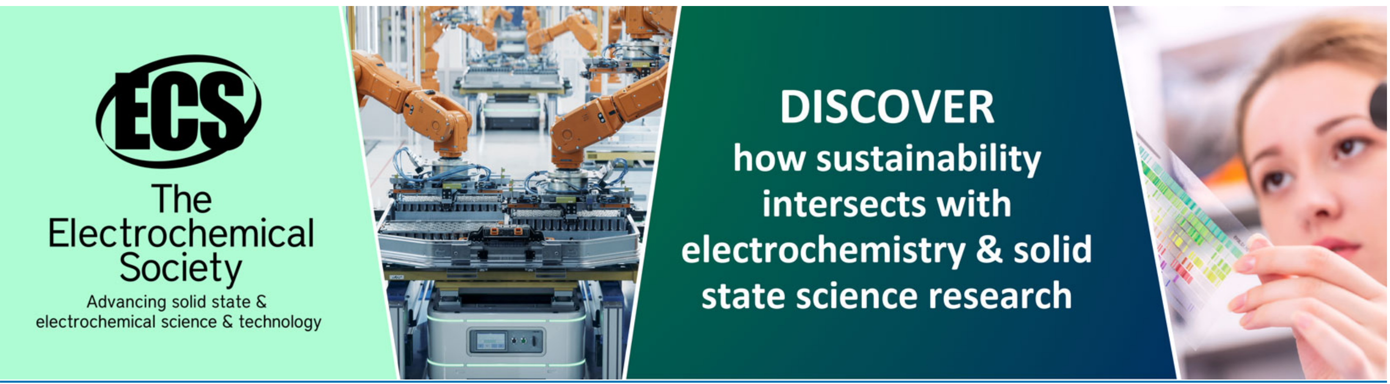
IOP Conference Series: Earth and Environmental Science

PAPER • **OPEN ACCESS**

Adaptation of segmented elbow on the flow meter by using CFD tools

To cite this article: K Panaś *et al* 2019 *IOP Conf. Ser.: Earth Environ. Sci.* **214** 012135



IOP Conf. Series: Earth and Environmental Science 214 (2019) 012135 doi:10.1088/1755-1315/214/1/012135

Adaptation of segmented elbow on the flow meter by using CFD tools

K Panaś[1,*], T Siwek[2], K Sztekler[2], W Kalawa[2], J Taler[3]

[1] EDF Polska S.A. Central Engineering, 31-574 Krakow, Poland
[2] AGH University of Science and Technology, Faculty of Energy and Fuels, 30-059 Krakow, Poland
[3] Cracow University of Technology, Head of the Chair of Machinery and Power Equipment, 31-864 Krakow, Poland

E-mail: krzysztofpanas@gmail.com

Abstract. This paper presents the possibility of non-invasive adaptation of the existing pipeline infrastructure in front of the condenser to the measurement of cooling water flow rate. In the analyzed variant, for the measurement of water flow rate in the cooling system, local pressure difference in the segmented elbow was applied. CFD (ANSYS CFX) was used as a tool for diagnosing flow structure, including the location of sampling points of pressure in the elbow. Many variants of flow were modelled ("What if?" analysis type) in this paper. These variants were used to determine the correlation between the differential pressure in the elbow and the flow rate. In addition, the article identifies the advantages and disadvantages of the solutions of this type against a background of standard elements and measuring distances and discusses the basic procedures for preparing and conducting CFD numerical experiment.

1. Introduction

Online measurement of condenser is important for the determination of the contamination level, as well as turbine and whole cycle efficiency [1,2]. The calculations are difficult due to the lack of precise steam parameters in the outlet from turbine. It is not possible to measure the steam dryness level, so it is extremely difficult to determine the steam enthalpy below the saturation line. The accurate flow measurement allows for balancing the condenser, from the cooling water side, without enthalpy of steam outcome from the last stage of turbine. The measurement of the cooling water flow rate is also of key importance for the heat-mass balance of cooling towers. Despite the importance of cooling water flow in the operation and optimization of the power unit, the measurement of this parameter with satisfactory uncertainty in a direct way is difficult, and sometimes impossible due to structural factors.

The metrological situation is complicated by the dimensions of the cooling water pipes and their layout – large diameters and the lack of long straights, which are defined and required in the majority of normative methods. Flow measurement using orifice plates and nozzles in such a situation is virtually impossible [3]. Stacking tubes are also not the best method of measurement due to the necessity of probing multiple points of the pipeline cross section, which generates problems with its mobility (lack of space, it is difficult to automate the measurement process).

In industrial practice, three types of measurement methods are most common for measuring the cooling water flow: ultrasonic flow meter, flow meters averaging dynamic pressure and elbow flow meters.

A measuring device for the ultrasonic flow measurement method consists of a measurement head and a central unit of the flow meter. Because of the measurement head arrangement and the need to maintain a coupling layer between it and the wall of the pipe, periodic inspections are required in order to replenish the coupling agent, control and secure the proper close contact of the head. Under certain conditions, the measurement with the use of this method is characterized by very good accuracy of approx. 1%. For such high accuracy, sufficiently long straight sections should be kept before measuring heads. Finding a straight section of the recommended length of 20 times the diameter on cooling water pipes of a large diameter is virtually impossible. When the condition of the straight line length is not preserved and a systematic measurement error is introduced, measurement accuracy drops to approx. 6% [14].

Flow meters that average dynamic pressure constitute another type of flow measurement used to determine the flow of cooling water stream. On the cross-section of the pipe, tubes averaging the total pressure are mounted. The measuring principle is based on the dynamic pressure medium recording, which is a function of the average speed of the medium flowing through the measuring section. In case of this measurement method, measurement uncertainties are at a level of approximately 3÷4% [14]. As in the ultrasonic method, the major role in obtaining an accurate measurement is played by sufficiently long straight lines before the measurement point.

The last method used are elbow meters. Such meters are typically used to measure the existing elbows by installing on the inner and outer surfaces of the elbow to define the occurring pressure differences. Measurement uncertainties in this type of measurement are at a level of 3÷4%. A considerable advantage of elbow measurement is that in large-scale industrial conditions, on such a large diameter pipe and without long straight lines, it is possible to achieve relatively high accuracy.

In view of the difficulties designating suitable locations for the measurement with ultrasonic methods or by using flow meters that average dynamic pressure, it is justified to choose an elbow flow meter as a way to measure the cooling water flow.

In this study, the possibility of non-invasive adaptation of the existing pipeline infrastructure in front of the condenser for the purpose of the cooling water flow measurement is analyzed. By using CFD (Computing Fluid Dynamics), the structure of the flow in the elbow segment of the pipeline supplying cooling water to the condenser was examined. Based on the numerical results, the dependence of differential pressure at selected probing points on the flow rate was characterized. It was proven that the existing elbows can be used as a reliable measuring instrument.

2. Analyzed facility

One of the elbows on the supply line of the infrastructure of the cooling water pipeline for the condenser in the second block of the Heat and Power Plant Krakow was selected. The selection was made based on the criterion of metrological suitability for balancing mass and energy of the whole plant. To eliminate the effect of boundary conditions on the results of the calculations for the elbow in question, numerical analysis covered the installation in front of and behind the elbow. It was particularly important to include calculations on the elbow located directly in front of the measurement elbow, because it has a significant influence on the structure of the flow in the test object. 3D geometry of the installation fragment was reconstructed on the basis of the available design documentation and facility inspection. The course and basic dimensions are presented in Figure 1.

The pipeline is in a horizontal position. Elbows are welded of five full segments and two halves. Segment curve geometry is adopted in accordance with KER-69/2.02 [4]. The stream of cooling water flowing through the analyzed installation part is 16400 m3/h (nominal value) at an operating pressure ratio of $2{,}837 \times 10^5$ Pa.

3. Numerical model

To describe fluid flow through the pipeline, two basic balance equations of fluid mechanics were used [5 to 9]. The continuity equation:

$$\frac{d\rho}{dt} + div(\rho\vec{v}) = 0 \tag{1}$$

The equation of momentum conservation (Navier-Stokes equation):

$$\rho\frac{d\vec{v}}{dt} = \rho\vec{F}_m + div\left[\left(-p - \frac{2}{3}\mu\, div(\vec{v})\right)I + 2\mu S\right] \tag{2}$$

where:

ρ - density of the fluid,
$\vec{v}$ - velocity vector,
F_m - unit mass force,
p - pressure
μ - dynamic viscosity of the fluid,
I - unit tensor,
S - tensor deformation speed [7].

Isothermal flow is assumed in the model. Water at a temperature of 25^0C is the working fluid, the density is constant ρ = const. (997 kg / m^3) and dynamic viscosity μ = const. ($8.899 \cdot 10^{-4}$ Pa·s). These assumptions lead to the simplification of transport equations in the following form:

$$div(\vec{v}) = 0 \tag{1.a}$$

$$\frac{d\vec{v}}{dt} = \vec{F}_m - \frac{1}{\rho} grad\, p + \vartheta\, \nabla^2\vec{v} \tag{2.a}$$

In equation 2.a., kinematic viscosity $\theta = \mu / \rho$ and the Laplace operator ∇^2 are introduced.

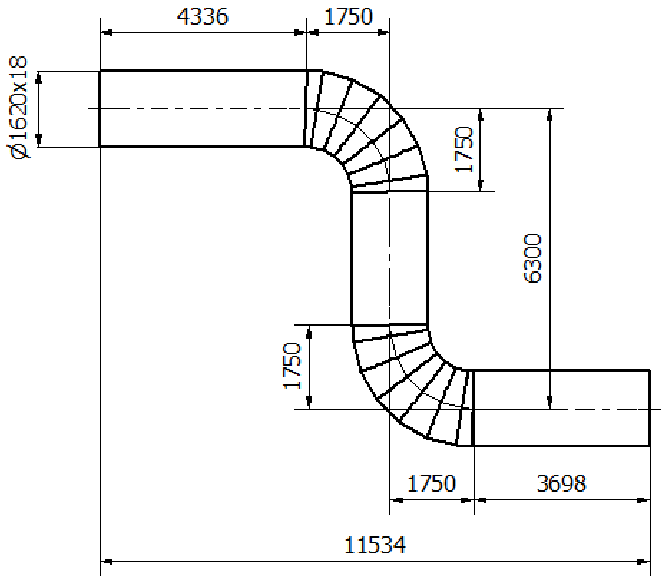


Figure 1. Modeled part of the cooling water installation.

Very dense computational grids and time-consuming solving (non-stationary calculations) are required for solving Navier-Stokes equations of motion in a direct way, so for DNS (Direct Numerical Simulations) calculations. For the analyzed case, DNS simulation is very difficult to implement - large dimensions of the object determine the use of enormous computer and license resources. In practical (industrial) calculations, the method of averaging variables referenced to time, leading to the RANS equations (Reynolds-Averaged Navier-Stokes equations) [5 to 9], is used most often. Assuming an average fixed turbulent motion of liquid, each of the flow parameters at the moment can be written as the sum of the average value and its fluctuations, respectively:

$$\begin{gathered} p = \overline{p} + p' \\ v = \overline{v} + v' \end{gathered} \tag{3}$$

To simplify a 3D model to a 2D model, assumed that the pipeline is parallel to the ground - XY plane in Figure 3 (gravitational acceleration is directed normally to the modeled plane direction, Z direction), and the flow is steady (no acceleration); the component of mass forces can be eliminated from equation 2a. To improve the accuracy of calculations, the pressure distribution in the cross section, which is a consequence of gravitational forces, has been analyzed. Pressure value in the mid-cross section height was used to correct the real 2D model. The calculation results of pressure distribution are shown in Figure 2.

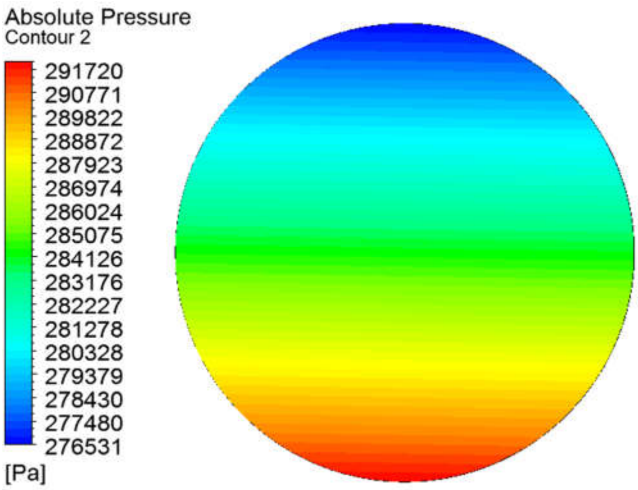


Figure 2. Absolute pressure gradient in the cross-section of the pipeline - the influence of the gravitational field.

Finally, after eliminating the forces of gravity, the equations 1.a and 2.a. in the indicator notation for the average flow medium determined take the form of:

$$\frac{\partial \bar{v}_i}{\partial x_i} = 0 \tag{1.b}$$

$$\frac{\partial}{\partial x_j}\left(\rho \bar{v}_i \bar{v}_j\right) = -\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i}\right)\right] - \frac{\partial R_{ij}}{\partial x_j} \tag{2.b}$$

Where R_{ij} is the Reynolds stress tensor related to an average rate of momentum change due to turbulent pulsation. In the Cartesian coordinate system, the tensor has the following form:

$$R_{ij} = -\rho\overline{v_j' v_i'} = \begin{bmatrix} -\rho\overline{(v_x')^2} & -\rho\overline{v_x' v_y'} & -\rho\overline{v_x' v_z'} \\ -\rho\overline{v_y' v_x'} & -\rho\overline{\left(v_y'\right)^2} & -\rho\overline{v_y' v_z'} \\ -\rho\overline{v_z' v_x'} & -\rho\overline{v_j' v_i'} & -\rho\overline{(v_z')^2} \end{bmatrix} \tag{4}$$

The system of equations 1.b and 2.b is not a closed system. In the Reynolds tensor, there are no six equations for individual components (equation 4). In this paper, the problem of additional equations is solved by calculating the impact of turbulence in Boussinesq approach [5 to 9]. According to his hypothesis, there is a relationship between the components of the Reynolds tensor and the deformation speed tensor. In the indicator notation, this relationship is expressed by equation 5.

$$R_{ij} = -\rho\overline{v_j' v_i'} = \mu_T\left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i}\right) \tag{5}$$

Turbulent viscosity $\mu_T$binds stresswith velocity field. This is a scalar function (often nonlinear) of multiple variables, such as physical and chemical properties of liquid or the nature of the flow. There are many empirical/semi-empirical mathematical models that solve the problem of turbulent viscosity.

Many of them have been adopted for commercial computer codes. As a complement to RANS equations, a two-equation turbulence model SST k-ω developed by Menter [11,12] is adopted for the concept. SST (shear stress transport) model is a hybrid model that combines the best characteristics of the k-ε and k-ω models. k-ε model works well in a free flow; however, k-ω model has a much better impact on flow modeling in the boundary layer. Additionally, SST introduces an element which limits "overproduction" of the kinetic energy of turbulence in areas of strong positive pressure gradients - a feature especially desirable in the flow of the sudden change of direction, such as the analyzed system of two 90º elbows.

Boundary conditions in Figure 3 are assumed for the mathematical model, taking static pressure of fluid on the outlet from the domain ($2{,}837\times10^5$ Pa ± correction for 2D model) and speed on the wall $v_{x,y,z}=0$. To define dependence characteristics of pressure drop in the control nozzles on the volume flow, the boundary condition at the inlet of the domain was specified as conditional on the external parameter (normal speed for the inlet port ($v_x = 0 \div 3$ m/sec). In the 3D model, the gravitational field is taken into account - the acceleration in the direction of "z" axis is equal to the gravitational acceleration. In addition, the symmetry condition on the surface of the longitudinal section is introduced in the 2D model. The experiment was controlled using ANSYS Design Exploration in the "what-if" procedure [8].

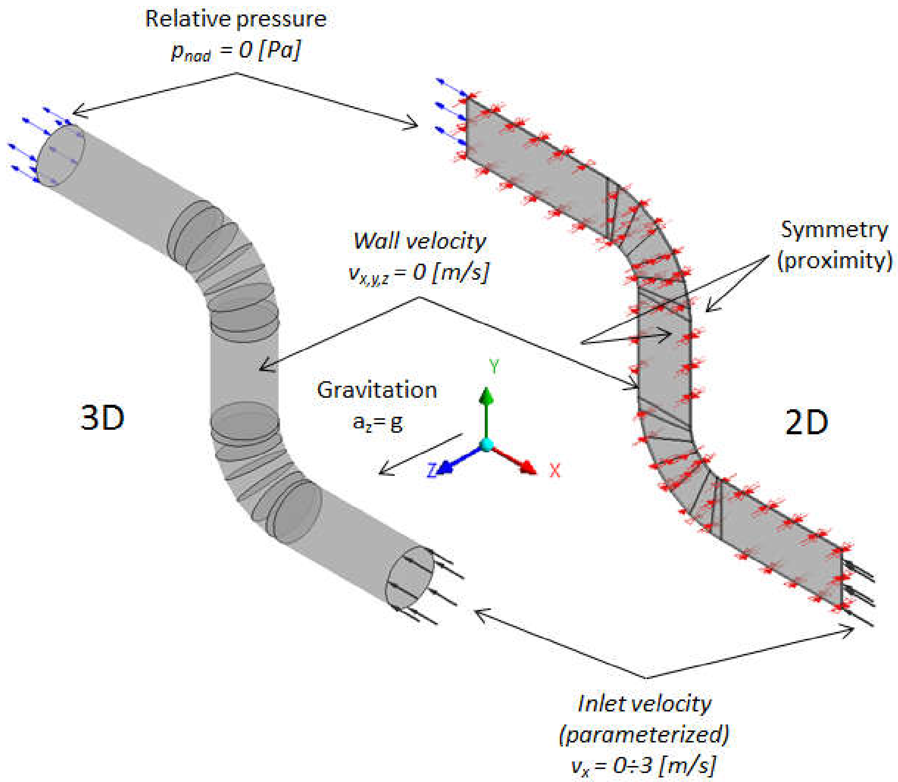


Figure 3. Model flow - boundary conditions.

The consequence of considerable dimensions of the analyzed object (internal diameter of the pipeline: 1.584 m) and restrictions on the size of the elements resulting from the turbulence model is a necessity to generate numerical grids of a very large number of components (several million). In order to reduce computation time and computer resources involved, the 3D model was simplified for 2D analysis. Comparative calculations of both models were made on a coarse grid, assuming heights of the elements on the pipe wall of a size equal to 50 mm. In both cases, the high quality structural grids were generated. The number and shape of the grid are presented in Figure 4.

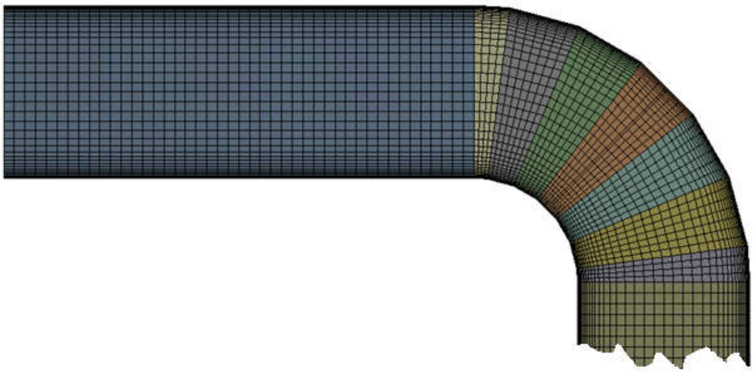


Figure 4. Fragment of a generated structural grid.

The mesh was generated by using ANSYS Meshing with the Multi Zone method. In the boundary region, according to the guidelines of the computation methods [7 to 9], 15-layer inflation was defined, with increment coefficient of subsequent layers equal to 1.2.

Figure 5 juxtaposes the results obtained for the 3D and 2D model in the form of the differential pressure depending on the flow. Additionally, Figure 6 identifies the places where differential pressure is measured. These locations were selected according to the assumption of identifying points/areas of minimum and maximum pressure in the flow of the elbow.

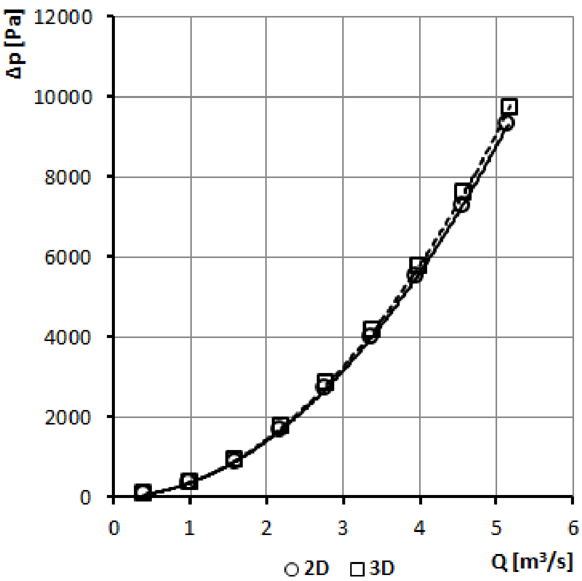


Figure 5. Comparison of the preliminary calculations for the 3D and 2D model.

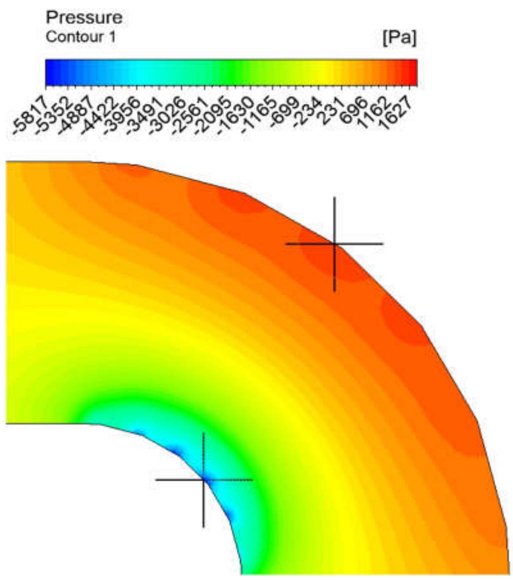


Figure 6. Identification of the pressure sampling locations.

The simulation and the results confirm the correctness of the assumptions for the simplification of spatial analysis for 2D calculations. Consequently, it is possible to generate a more precise grid in boundary areas (smaller components), which directly translates intoquality and time of calculations. In the 2D model, sensitivity of calculations to the density of the numerical grid was analyzed. The test results are shown in Figure 7.

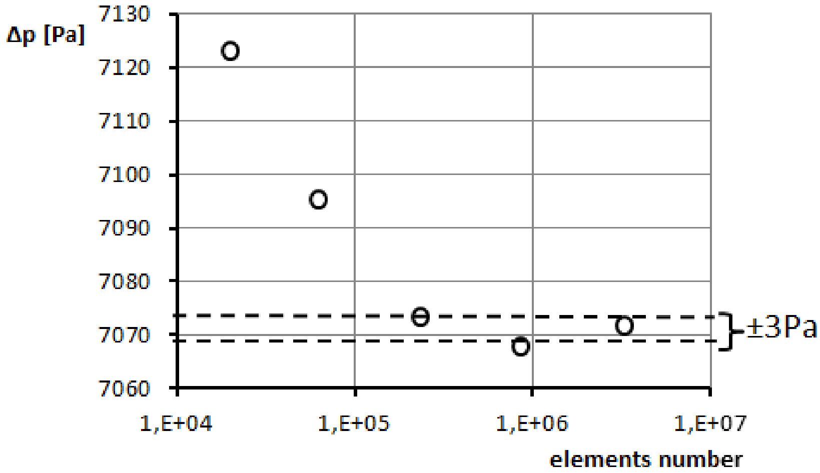


Figure 7. Examination of the susceptibility of solutions to the size of the numerical grid.

The analysis of the sensitivity of the results to the size of grid elements shows that more than a quarter million (0.25 x 10^6) grid elements do not significantly affect the results of mathematical calculation. For such a dense grid, the smallest elements on the walls (layer inflation) have a height of about 0.5 mm, which guarantees the fulfillment of the condition of the value of parameter Y+ in the SST turbulence model in the whole range of the analyzed speeds [10].

4. Calculation results

Figure 6 presents the location of pressure sampling points for the purposes of assessing the correctness of the adopted model. In an industrial environment, it is not possible to determine sampling points for pressure. In addition, the initial location contributes to a high risk of underestimation of pressure due to the location of welding and assembly inaccuracies at the junction of two segments. In a further numerical experiment, the pressure pulse sampling was performed with two simulated measurement points. The proposed location of the nozzles was in the half of the middle segment of the second pipe elbow, on two opposite sides. Nozzles are threaded tubes having an internal diameter of 20mm, and the axes of which lie in the half of the diameter of the main pipe (within "z" axis).

The results of the simulation of the numerically optimized model are presented in Figure 8 and 9. Figure 8 shows the comparison of two characteristics of a differential pressure flow. One function applies to the maximum possible pressure differences achieved at the elbow, the other refers to the proposed measurement points.

There was no significant decline in the quantity of the measured value. The suggested location of pressure intake nozzles may be accepted for industrial purposes. For easier description of the flow measurement device, it is a common practice to define a flow meter constant. The equation for the average flow rate according to the pressure differential is as follows:

$$v = K \cdot \sqrt{\frac{2 \cdot \Delta p}{\rho}} \tag{6}$$

where K-factor should be a fixed value for the primary element in the whole range of the measured flow. K-factor was determined by linear regression on the basis of the relation shown in Figure 9. For the presented pipe elbow with pressure measurement in proposed nozzles, flow factor K is 0.689.

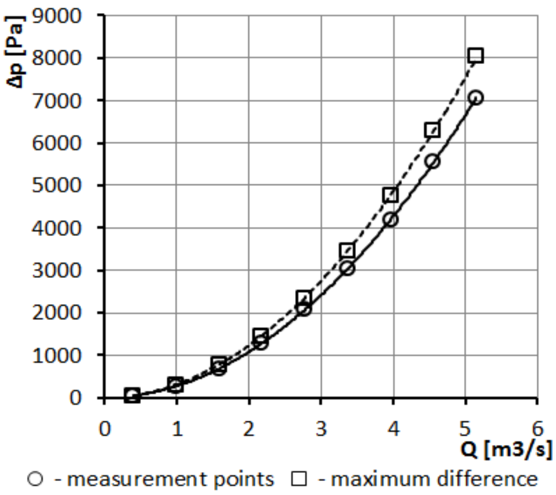


Figure 8. Dependence of the differential pressure on the elbow on the flow.

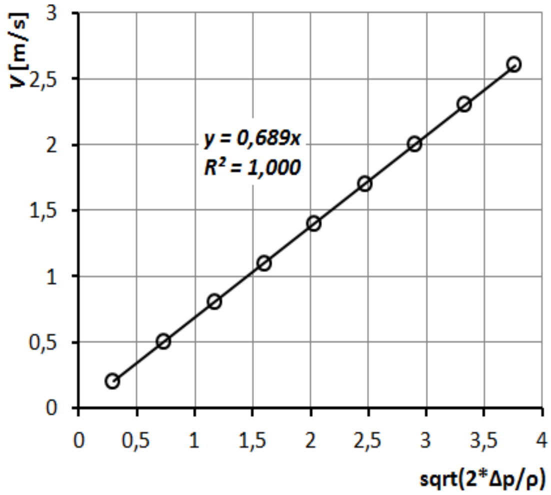


Figure 9. Linear regression - the estimate of K.

The statistics of the experiment show that the proposed method of linear regression gives a very good representation. The value of the designation R^2 is equal to ≈1, which confirms the existence of a perfect correlation in the samples. The maximum uncertainty of the estimated factor K is $\pm 1 \cdot 10^{-3}$.

Figure 10 shows the quality results for flow through installations for the nominal capacity of (16400 m^3 / h). Figure 10a presents the pressure distribution in the longitudinal section of the pipeline, Figure 10b - flow velocity. The presented results show that the pipe elbows influence each other and the results cannot be easily transferred to another flow system.

a) b)

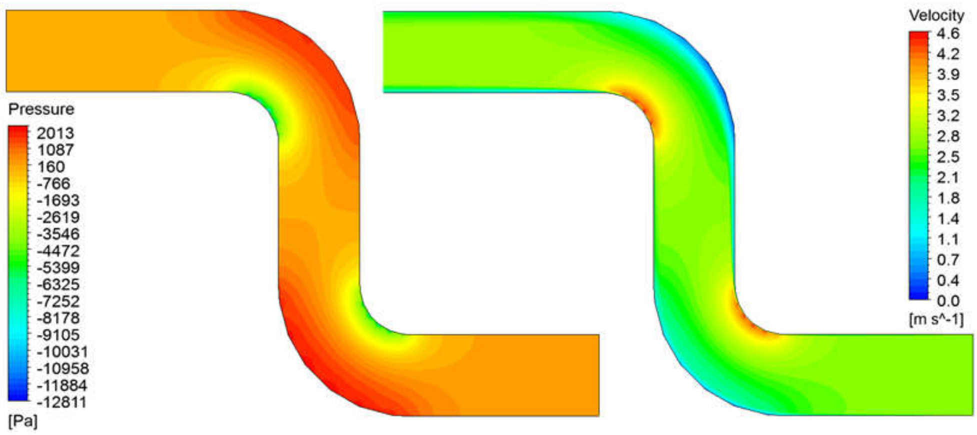


Figure 10. The results of CFD simulation a) the dependence of the differential pressure on the elbow *on the* flow, b) velocity.

5. Summary

The presented study confirms the possibility of adopting the existing pipeline infrastructure (especially pipe elbow elements) for flow rate measurement, using CFD methods.

It is important to prepare the numerical model properly. This includes generating a corresponding numerical grid and the adoption of appropriate initial and boundary conditions. Due to limitations in computing power, it is necessary to simplify the model, while maintaining the essential features of a real object. Narrowing simulation exclusively to one pipe elbow and taking too small distances results in serious model errors.

The modeling of the concept in question was based on the experience gained within the research work conducted at the Department of Thermal and Fluid Flow Machines of AGH University of Science and Technology in Krakow, including these presented in the unpublished work [13], in which standardized Venturi measurement tube was modeled. The result of Venturi tube simulation flow was verified by bench tests. The CFD model showed a very good correlation with the real flow, as shown in Figure 11.

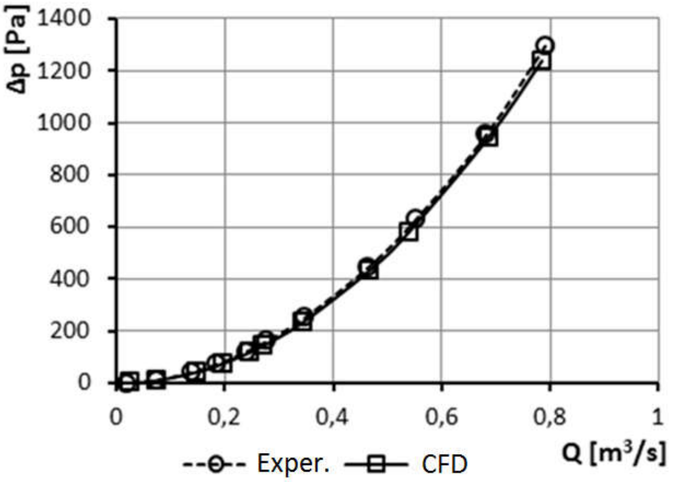


Figure 11. Verification of Venturi tube numerical model by experimental results [13].

CFD studies play an increasingly important role in industrial applications. They reduce time required to solve the problem and the associated costs. Results generated from numerical models can be successfully transferred to applications in real systems.

Acknowledgments

The article was funded from government money Faculty of Energy and Fuels number 11.11.210.376.

References

[1] Laković M S, Laković S V, Banjać S M, 2012 *Thermal Science* Vol.**16** Suppl. 2 pp. 375÷385
[2] Rutberg M J, 2012 Master Thesis Massachusetts Institute of Technology USA (on-line: http://hdl.handle.net/1721.1/74674)
[3] Gondek A, 2010, *Differential pressure flowmeters*, Publishing House of Cracow University of Technology
[4] Catalog of Pipeline Elements KER-69/2.02
[5] Cebeci T, Shao J, Kafyeke F, Laurendeau E, 2005 *Computational Fluid Dynamics for Engineers*, Springer Heidelberg Germany543210
[6] Roache P J, 1998 *Fundamentals of Computational Fluid Dynamics*, Hermosa Publishers
[7] Kazimierski Z, 2004 *Fundamentals of fluid mechanics and numerical flow simulation*, Publishing House Lodz University of Technology
[8] ANSYS Customer Portal 2016 (on-line:https://support.ansys.com)
[9] ANSYS CFX-Solver Theory Guide 2011, Release 14.0
[10] Ariff M Salim M S, Cheah S C, 2010 *Progress in Computational Fluid Dynamics*, Vol.**10** Nos.5/6, pp. 341÷351
[11] Menter F R, 1993 24th Fluid Dynamics Conference Orlando Florida AIAA Paper 93÷2906
[12] Menter F R, 1994 AIAA Journal, Vol.**32**, No 8. pp. 1598÷1605
[13] Sarzyński Ł, 2012 Engineering Thesis AGH Krakow
Andruszkiewicz A, Kubas K, Pliszka P, Wędrychowicz, 2014 Measurement Automation Robotics PAR No.**9** pp. 91÷95