

The hydraulic and numerical analyses of the operation of the control valve in the central heating system

J Rogula¹, P Szulc¹

¹Wroclaw University of Science and Technology, Faculty of Mechanical and Power Engineering, Department of Design Fundamentals and Fluid–Flow Machinery, ul. Wybrzeze Wyspianskiego 27, 50–370 Wroclaw, Poland

E-mail: janusz.rogula@pwr.edu.pl

Abstract. The article presents the results of experimental and numerical tests of a surface heating flow control system consisting of a distribution rail along with the fixed valve. The components were measured, their solid models were prepared and used to make a flow model. In a flow model geometry the measurements of the test stand elements were considered in order to compare the results.

1. Introduction

In a surface heating the whole area of a floor, wall or ceiling of the room [2] is a heater. It is a kind of a low temperature heating where the medium (water or glycol) flows through the spread on the plane built-in installation and heat exchange takes place predominantly on the basis of radiation. An advantage of this type of heating system is a possibility of spreading out the installation on very large surfaces in a few or several independent branches with the ability to control the heating of each branch from one central point. It allows to simplify the way of regulation.

The efficiency of a surface heating system and the accuracy of temperature regulation in a room depends on the quality of control valves fixed to a distribution rail (Figure 1). Up to several control valves can be fixed on one rail [3].

Warm water under pressure flows through the rail. It is then divided into particular streams flowing through the branches. The quantity of water in the branches is controlled by the valves fixed in a rail. Control with valves causes the increase of local resistance in a stream of liquid.

Knowledge of the flow resistance through the determination of the valve coefficient k_v , which can be defined on a test stand as well as numerically, by means of a CFD software, will help to specify pressure losses in a heating system and choose an appropriate pump.

2. Experimental research

2.1. Object of study

A set consisting of a brass distribution rail and a flow control valve plug were applied for the analysis concerning the operation of control valves of the surface heating. The elements presented in Figure 1 were assembled, connected to the test stand and further tested. The construction of a distribution rail and a valve were analyzed.



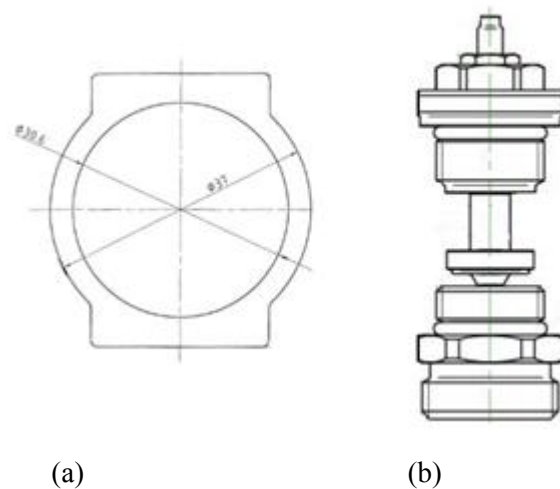


Figure 1. Object of analysis: (a) distribution rail cross-section, (b) model valve.

A distribution rail 155 mm long, of an inner diameter $\phi 30,6$ mm, and outer $\phi 37$ mm makes it possible to connect three control valves in a serial configuration. Three threaded holes were made in a rail G1/2 which allow to fix three enabling to put three branches. Moreover, two connectors G1 were made in the rail. One connector is used for fixing a pipeline with the supplied water and the second one allows to join another connecting rail. A valve seat ring was fixed to one side of the rail, on the other side there was the body with a fixed spindle and a valve plug of a diameter $\phi 17$ mm (Figure 2). The valve plug (4) was made in a form of a vessel where the sealing was located (5) $\phi 15$ mm. The task of the sealing was to eliminate any leakage in a closed position, i.e. when the valve plug touches the seat ring (6). Maximum lift of the valve plug amounts to 2.5 mm which at the same time characterizes the level of the valve regulation. A spindle (1) of $\phi 8$ mm was used for flow throttling. The valve spindle for tests was driven by means of a handle connected to the pin sealed with an O-ring in body stuffing box (3). The force deforming the spring (2) regulates the pressure of the valve plug to the seat ring.

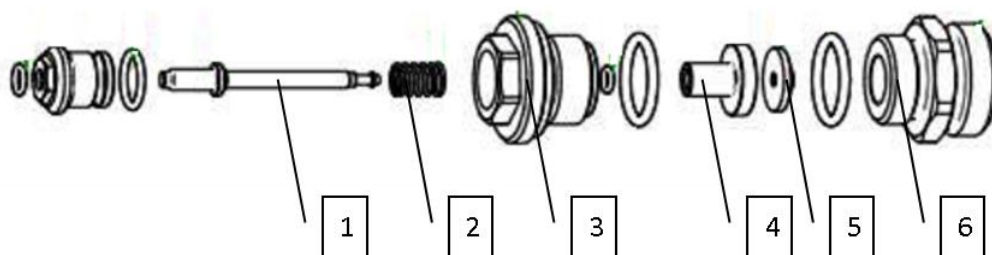


Figure 2. Model valve, dedicated for tests: 1– spindle, 2 – spring, 3 – body, 4 – valve plug, 5 – elastic insertion, 6 – seat ring.

In order to prepare the distribution rail for tests one of the connectors as well as two pairs of the outlet holes were covered.

2.2. Test stand and measurements methodology

The tests of flow resistance through the valve were carried out on a test stand dedicated for testing central heating flow elements. The test stand was shown in Figure 3.

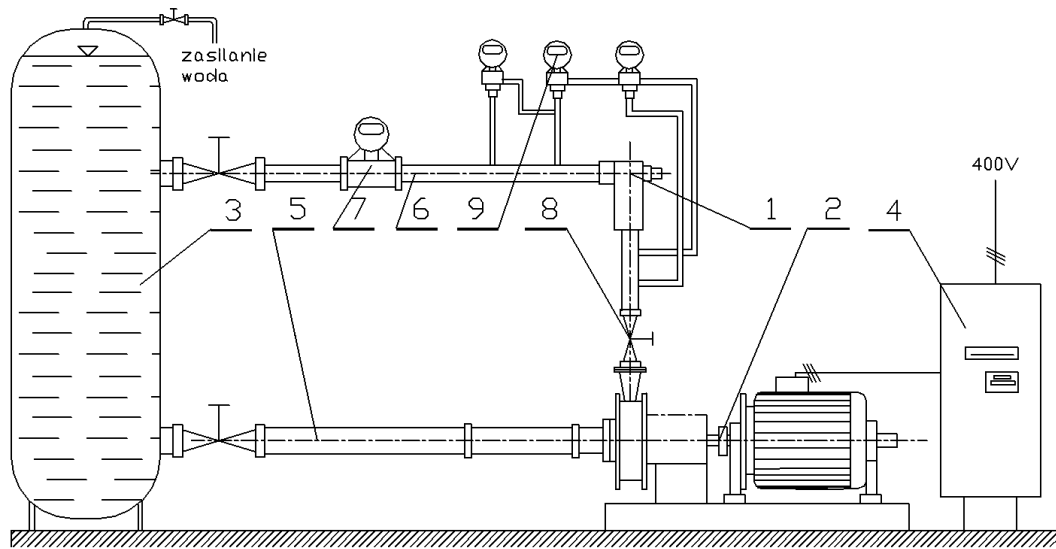


Figure 3. Test stand for testing flow elements of central heating systems: 1 – rail with tested valve, 2 – pump, 3 – tank, 4 – frequency converter, 5 – suction pipe, 6 – discharge pipe, 7 – electromagnetic flow meter, 8 – control valve, 9 – manometers.

Water flow on a test stand took place in a closed system. The measuring system included a rail with one valve (1) and a flow in the system was forced by means of a centrifugal pump (2). Since it was necessary to regulate pump's performance a frequency converter (4) was used in the measuring system. The feeding element was a closed tank (3) from which the pump sucked the medium with a suction pipe (5) and drained it with a discharge pipe (6). The test stand was also equipped in a control valve (8) throttling the flow before the tested valve. The flow through the valve was measured by means of an electromagnetic flowmeter (7) in grade 0,2 ensuring standardized lengths of pipelines before and after the place of measurement; due to flow disorders caused by the valve work the places of measurement were located in the distance of 15 diameters before and after the valve. The measurement of static pressure was realized by means of an electronic manometers (9) in grade 0,2. In order to compensate the pressure losses in straight pipe fractions the loss on equivalent to the length and diameter pipelines fractions was measured: feeding and return. The temperature of the medium was determined by means of a mercury thermometer. The pumped medium was pure water supply of density equal 999 kg/m^3 . While taking measurements of parameters necessary to determine basic characteristic of the valve constant feeding conditions were maintained. The measurements were taken in the steady state.

To determine characteristics of the change of valve coefficient k_v as a function of the number of the regulation wheel rotations it was necessary to alter performance of the pump forcing the flow through the tested valve. It was assumed, in accordance with norm [1] that on the valve, regardless of its setting the drop of pressure equal 1 bar would be kept. To enable the realization of pressure drop condition on the valve, rotations of the pump shaft were changed and locally regulated with a control valve (8) for a given setting of the regulation wheel of the tested valve.

To determine the flow coefficient k_v the following formula was used:

$$k_v = \frac{Q}{\sqrt{\Delta p}} \quad (1)$$

where:

Q – volumetric flow,

Δp – pressure drop on the tested valve.

2.3. Results of the real research

Experimental research included determination of the flow coefficient k_v in course of the valve opening and closing. The results can be found in Figure 4.

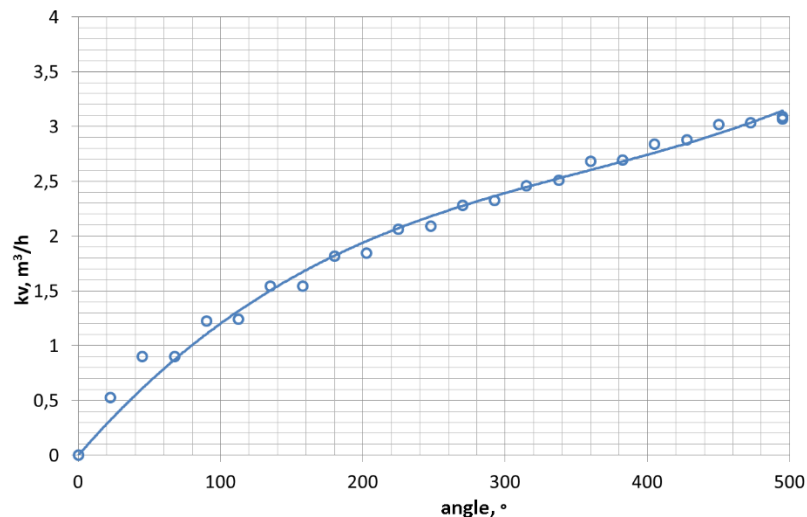


Figure 4. Characteristic of the valve fixed in a rail $k_v = f$ (rotation angle).

While analyzing the curve course it should be noticed that for a full opening of the valve the coefficient k_v equals about $3.1 \text{ m}^3/\text{h}$. The curve of changes k_v in the whole course of changes of the control valve rotation angle is increasing. Due to steepness we can divide the characteristic into two parts. To the opening angle value 150° the increase k_v equals $1.13 \text{ m}^3/\text{h}$ for 100° of the valve pin rotation, however, more than 150° only for $0.37 \text{ m}^3/\text{h}$ for 100° .

3. Numerical simulations

3.1. Computational model

Computational Fluid Dynamics makes it possible to conduct numerical tests of the real objects with a view to checking the accuracy of construction calculations as well as explaining and providing the reappearance on accompanying phenomena [4,5]. Due to CFD, reducing investment costs, we are able to explain a lot of phenomena, particularly when the real measurement methods may lead to, e.g. devastation of the studied object.

The three dimensional flow geometry model subjected to simulation was presented in Figure 5. To precisely reflect the physical phenomena the 3D model constituted a carbon copy of a real model. It was decided to carry out flow simulations for three chosen settings of the valve. It was assumed that the valve plug lift would be equal 0.8 mm (Figure 6, 7), 1.6 mm (Figure 8, 9) and 2.4 mm (Figure 10, 11).

The process of model discretization was conducted by means of Ansys Icem CFD software by making a non structural grid (tetrahedral). A prismatic grid was applied close to the walls in areas particularly sensitive to flow disorders, i.e. valve areas. This allowed to secure the value y^+ during the calculations below 1.5.

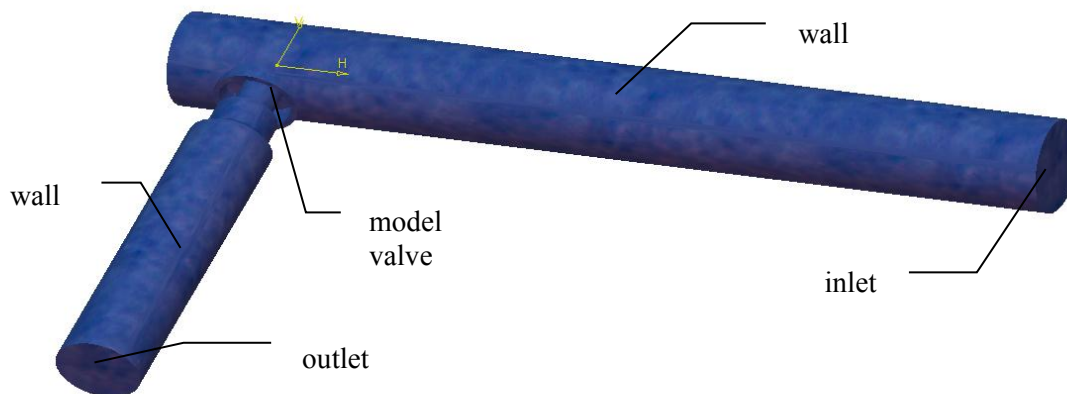


Figure 5. Three dimensional model of the flow through partly open valve fixed in a distribution rail subjected to CFD simulations.

Numerical research of the investigated object was conducted based on commercial software Ansys Fluent CFD. The software enables, with the use of the finite volume method, iterative solution of momentum, energy and mass maintaining equations. Calculations were performed by means of a turbulence model k- ω SST and changeable geometric parameters of the valve. Numerical analysis was carried out as stationary, a working medium was pure water. Convergence of the results for all equations was obtained at the level 10^{-5} . In the analyzed configuration the value of static pressure as well as estimated turbulence level were assumed at the inlet and outlet. An output parameter was the flow Q . The discussed numerical model was divided by planes for a better visualization of the phenomena taking place in the channel. The distance between the planes was chosen so that, at the same time, it was possible to show the swirls of the flow. As a result of the carried out numerical simulations data enabling the analysis of a stationary flow through the tested system of the valve fixed in a distribution rail were collected.

3.2. Qualitative results of numerical simulations

The prepared computational models enabled to perform the qualitative identification of the flow in the investigated flow system.

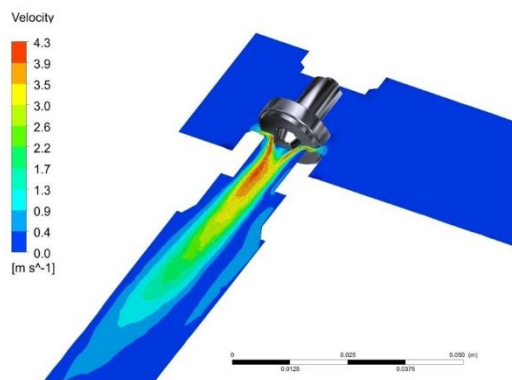


Figure 6. Water velocity distribution in the valve, valve plug lift 0.8 mm.

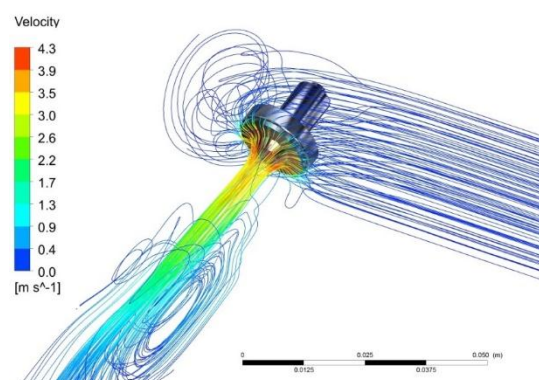


Figure 7. Tracks of water particles in the valve, valve plug lift 0.8 mm.

Figure 6 presents flow velocity distribution in the plane going through the axis of the distribution rail as well as pipe draining the liquid from the valve. Figure 7, on the other hand, presents the lines of liquid particles course for the gap between the valve plug and a seat ring equal 0.8 mm.

While analyzing the obtained results one should notice a zone of the increased velocity which occurs directly between the valve plug and the seat ring. The gap between the flat surface of the seat ring and the flat surface of the valve plug decides about the velocity. The biggest velocity occurs directly under the valve plug. The occurrence of the dead area directly under the elastic insertion of the valve plug and sharp edges of the seat ring influencing the flow decide directly about the localization of the biggest velocity. The received flow picture can be improved by the application of the valve seat ring construction with removed sharp edges on the flow way, e.g. with the rounded edges or of a bell-mouth cross-section. The zone of the non uniform flow behind the valve plug leans left which is the result of the impact on the vortex zone, visualized through the comparison of Figure 7, Figure 9 and Figure 11, where the tracks of water particles were shown at a different location of the valve plug in relation to the seat ring. The vortex zone, which causes the dead zone, fills about a half of the flow channel. It causes the increase of local velocity, therefore, the rise of flow loss. This area is localized on the internal side of the elbow which is a standard picture of the flow while maintaining a constant winding [6]. Its existence is also connected with a violent extension of the pipe, and the flow in the elbow blocks its propagation in a reverse way.

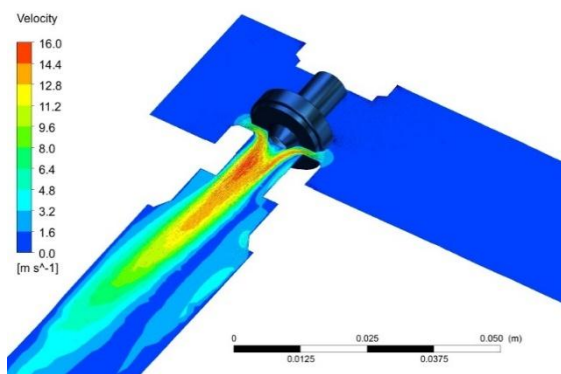


Figure 8. Water velocity distribution in the valve, valve plug lift 1.6 mm.

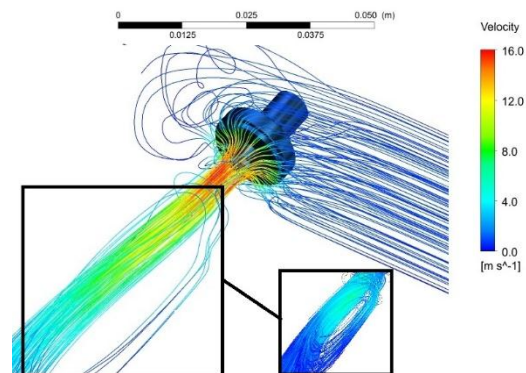


Figure 9. Tracks of water particles in the valve, valve plug lift 1.6 mm.

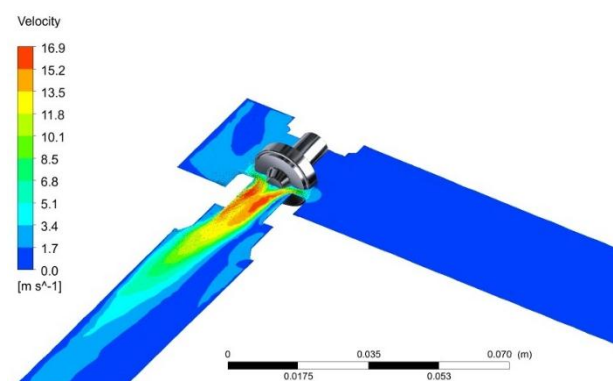


Figure 10. Water velocity distribution in the valve, valve plug lift 2.4 mm.

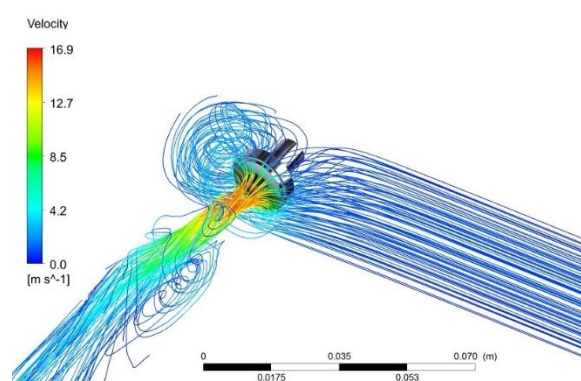


Figure 11. Tracks of water particles in the valve, valve plug lift 2.4 mm.

3.3. Quantitative comparison of the results of real and numerical research

The performed numerical simulations made it possible to conduct a comparative quantitative analysis of the flow through the investigated valve. Therefore, the results of the real research, obtained on a test stand, were compared with the results of CFD calculations. The results can be found in Figure 12.

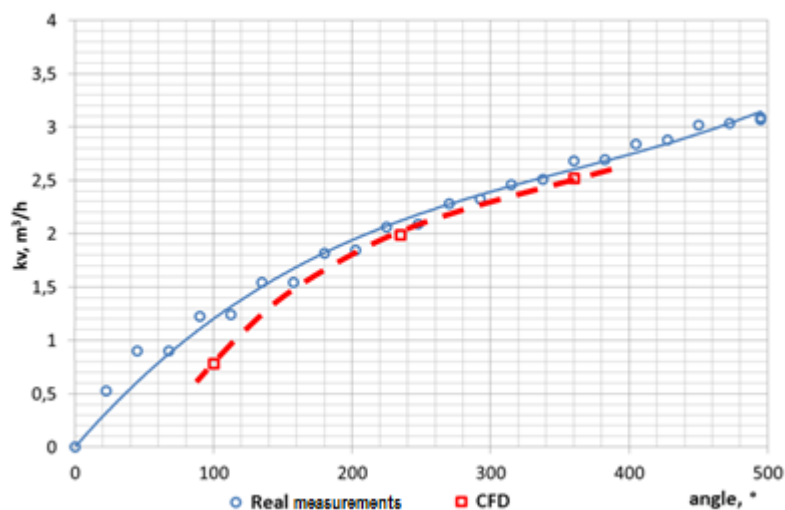


Figure 12. Comparison of numerical simulations results with measurements.

While analyzing the chart one should notice that discrepancy between the two research methods is small. Its maximum value equals $0,4 \text{ m}^3/\text{h}$ for the rotation angle of a control wheel equal 100° and it decreases along with opening the valve. The minimum value was obtained for the rotation angle 360° and it was about $0,1 \text{ m}^3/\text{h}$. The courses of two characteristics are close in the range of a rotation angle exceeding 240° . Below this value the curve obtained through CFD calculations is characterized by a higher steepness, which causes approximately linear increase of the difference between the two research methods. It should be noticed that all result points for numerical simulations lie below the curve defined by the real research. This might be caused by the fact that the computational model does not include the influence of time on the obtained results and it is also important that along with the reduction of the rotation angle the height of the gap between the valve plug and seat ring diminishes and modeling of the flow in the gaps is currently quite problematic and often the obtained results are far from reality.

It should be concluded that the conducted numerical simulations are close enough to the real physical phenomena accompanying the liquid flow through the valve. It allows to implement those methods in a process of the valve construction optimization, variant analysis of the solution of its flow hydraulics as well as conducting the process of its optimization. It should be highlighted that although the accuracy of the CFD method is high each construction must undergo the real research in the final phase.

4. Conclusions

The obtained real research results related to the results of numerical simulations allowed to come to the following conclusions:

- methods of flow simulations can be successfully applied to determine the valve flow coefficient k_v ;
- discrepancy between the CFD simulations and the real research increases along with the reduction of the valve opening angle. This being caused by the change of the solid body height limited by the seat ring and the valve plug surface and the necessity of flow simulations in narrow gaps;
- presented velocity field as well as tracks of water particles showed a considerable non uniformity of the flow behind the valve;

- the swirl occurring behind the valve is inadvisable;
- for the value of lift 2.4 mm the stream of liquid has swirls after flowing through the area of - valve plug - seat ring;

Currently the authors are working on the experimental and numerical analyses of the cooperation among the three valves fixed on one rail.

References

- [1] PN-EN 60534-2-1 *Przemysłowe zawory regulacyjne. Wydajność przepływowa. Równania wymiarowania zaworów do przepływu płynów w warunkach instalacji*
- [2] Bean R 2009 History of Radiant Heating and Cooling Systems *ASHRAE Journal*
- [3] Chapuran A 2009 Radiant idea *Technology-Hydronic Heating*
- [4] Davis J and Steward M 2002 Predicting Globe Control Valve, Performance—Part I: CFD Modeling *ASME*
- [5] Sonawane V and others 2012 Design and Analysis of Globe Valve as Control Valve Using CFD Software, *Journal of Mechanical and Civil Engineering*
- [6] Ferrari J and Leutwyler Z 2008 Fluid flow force measurement under various cavitation state on a globe valve model *Proceedings of 2008 Pressure and Vessels Piping ASME conference*