ITM Web of Conferences **15**, 07011 (2017) DOI: 10.1051/itmconf/20171507011
CMES'17

Optimisation of liquid flow in cavitation tunnel using CFD method

Robert Jasionowski[1,*], and *Waldemar* Kostrzewa[1]

[1]Maritime University of Szczecin, Institute of Basic Technical Sciences, 2-4 Willowa Str., 71-650 Szczecin, Poland

Abstract. Computational Fluid Dynamics CFD is a type of software using a numerical modelling (finite element methods and finite volume methods) based on Navier-Stokes equations. This approach allows determining distributions of pressure, velocity, temperature and other features of a flowing media. In the present work. CFD software was used to a development of three-dimensional model of a cavitation tunnel, used for an examination of cavitation resistance of structural materials, as well as to simulate a liquid flow through the tunnel. The simulation was carried out for various assumed flow rates, to determine an optimal value leading to a more extensive area of low pressure assisted by a cavitation phenomenon.

1 Introduction

A progress in computer-based methods in the 20th century has led to an increase of popularity of the CFD as engineering software applied in various industry branches (i.e. aviation, automotive, medicine, petrochemical, energy etc.). The CFD has been applied to simulate lamellar and turbulent flow of liquids, a gas transport in combustion engines, a gas flow upon designing of aircrafts and sea crafts. Furthermore, the CFD allows verifying a nature of interaction between a flowing medium and an involved solid body [1-6].

A development of complete flow analysis giving many parameters (e.g. a stream course, local pressure drops, changes in flow rates, temperature variations) must be preceded by:

- building three-dimensional model of the analysed component (including a proper materials selection);
- assessing geometry of flowing medium (including its physical and chemical properties);
- setting boundary conditions of the flow by considering all necessary parameters;
- defining nodes and a mesh density in the most crucial parts of the working space;
- setting of parameters of analysis and a number of iterations;
- discussing obtained results.

Furthermore, the CFD allows performing the same analysis for a few different scenarios, i.e. for various initial parameters or model geometries.

In the present paper, the results of analysis of liquid flowing through a cavitation tunnel, under various flow rates, were discussed. The main purpose of presented simulation was to optimise flow parameters, in terms of formation of the largest low-pressure area leading to a cavitation phenomenon. An establishment of optimal flow conditions will allow conducting cavitation resistance test in shorter time, as well as give information about the best distribution of measurement and controlling devices.

2 Numerical model

2.1 Physical model

In the present work, a cavitation tunnel (Fig. 1), being a part of cavitation resistance measurement device, has been subjected to the analysis.

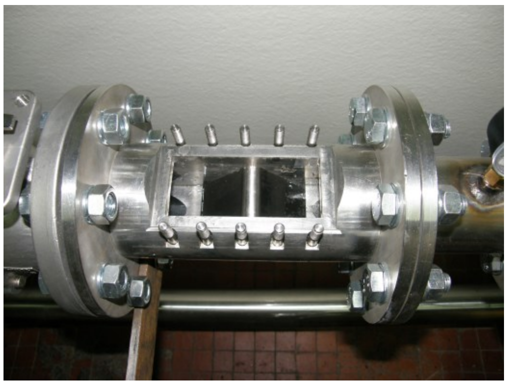

Fig. 1. Cavitation tunnel.

A PML2 80/200 type pump, supplied by 15kW engine, and LG iG5A inverter push water through the tunnel.

In previous experiments a rotor inside the PML2 80/200 type pump had been rotating with 2900 rpm, giving a flow rate of 9,95 m/s and delivery of a pump 200 m^3/h (measured in the tunnel inlet). The application of LG

* Corresponding author: r.jasionowski@am.szczecin.pl

 DOI: 10.1051/itmconf/20171507011

iG5A inverter allowed setting a freely chosen rotational speed of the rotor; thus, controlling the flow and delivery rate. The cavitation induction system in the tunnel consisted of barricade and counter-barricade systems, and was similar to those located in University of Hannover, Hiroshima University and Institute of Fluid-Flow Machinery of the Polish Academy of Sciences.

2.2 Flow field model and boundary conditions

The geometric model of cavitation tunnel was designed in AutoDesk Inventor Professional 2018 software, by using a dimensional accuracy of 0,1 mm (Fig. 2).

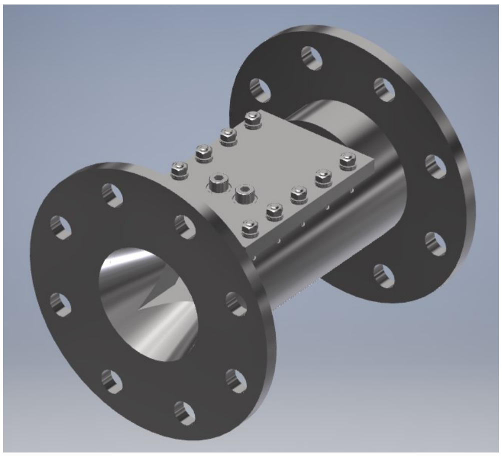

Fig. 2. The geometric model of cavitation tunnel.

The geometric model of cavitation tunnel was made of six own-designed components (body, cover, upper and lower barricades, two flanges and 53 components from pre-existed libraries (Fig. 3)).

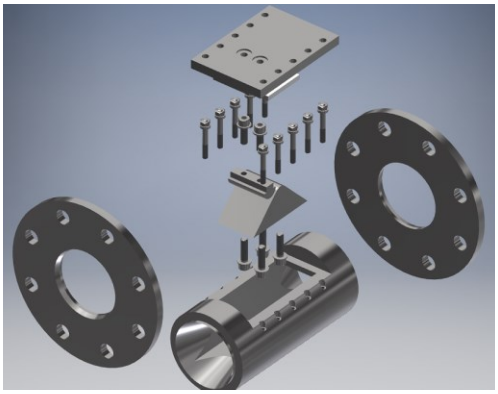

Fig. 3. The geometric model of cavitation tunnel - components.

The tunnel was made of 0H18N9 stainless steel, so the same material was also used upon the simulation of flow.

Volume mesh model (Fig. 4) and numerical analysis of liquid flow in cavitation tunnel were also designed in AutoDesk CFD 2017 software.

In order to increase the precision of measurements, tenfold concentration of mesh nodes between upper and lower barricades and under test sample (size adjustment parameter was 0,1) was performed. Additionally, components having no direct contact with the flowing liquid, as well as the roundness and chamfers of edges below 0,1 mm, were excluded from the analysis. Consequently, the achieved volume mesh model contained 960 000 nodes.

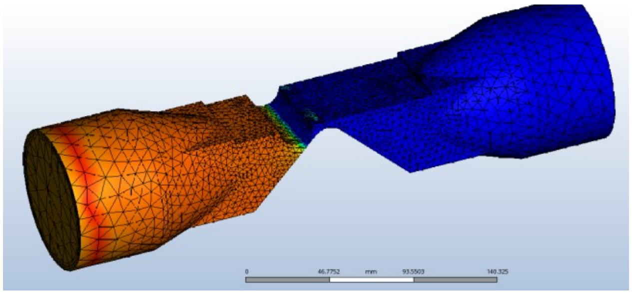

Fig. 4. Volume mesh model.

The following chemical and physical parameters of flowing water were used upon the analysis: density of 0,9982 g/cm3, viscosity of 0,001003 Pa·s, temperature of 25 °C, thermal conductivity of 0,0006 W/mm·K, bulk modulus of 2,18565 GPa.

The numerical analysis was carried out by using a liquid temperature of 25 °C and the following five different values of the velocity in the tunnel inlet: 7,74 m/s, 8,85 m/s, 9,95 m/s, 11,06 m/s and 12,16 m/s obtained for delivery rate: 160 m^3/h, 180 m^3/h, 200 m^3/h, 220 m^3/h and 240 m^3/h (maximum pump delivery rate).

3 Analysis of simulation and test results

The conducted analyses for five different velocity values allowed receiving a distribution of flow rate and pressure inside the cavitation tunnel.

To precisely determine positions of low pressure areas, additional 7 measurement points were added to the tunnel axis. The points separated by 5 mm distance were placed parallelly to the surface of sample at the distance of 0,5 mm (Fig. 5).

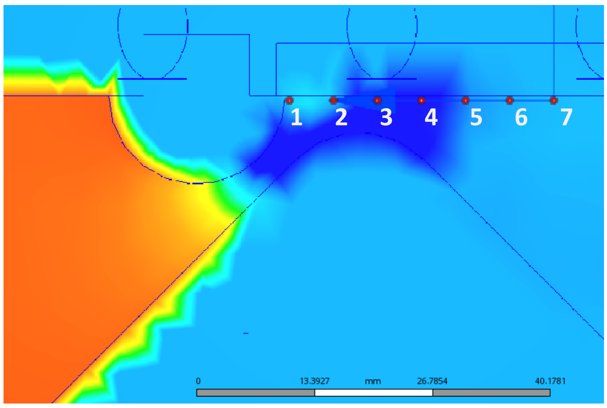

Fig. 5. Measurement points.

The results of the velocity are summarised in the Fig. 6, and pressure results in the Fig. 7.

DOI: 10.1051/itmconf/20171507011

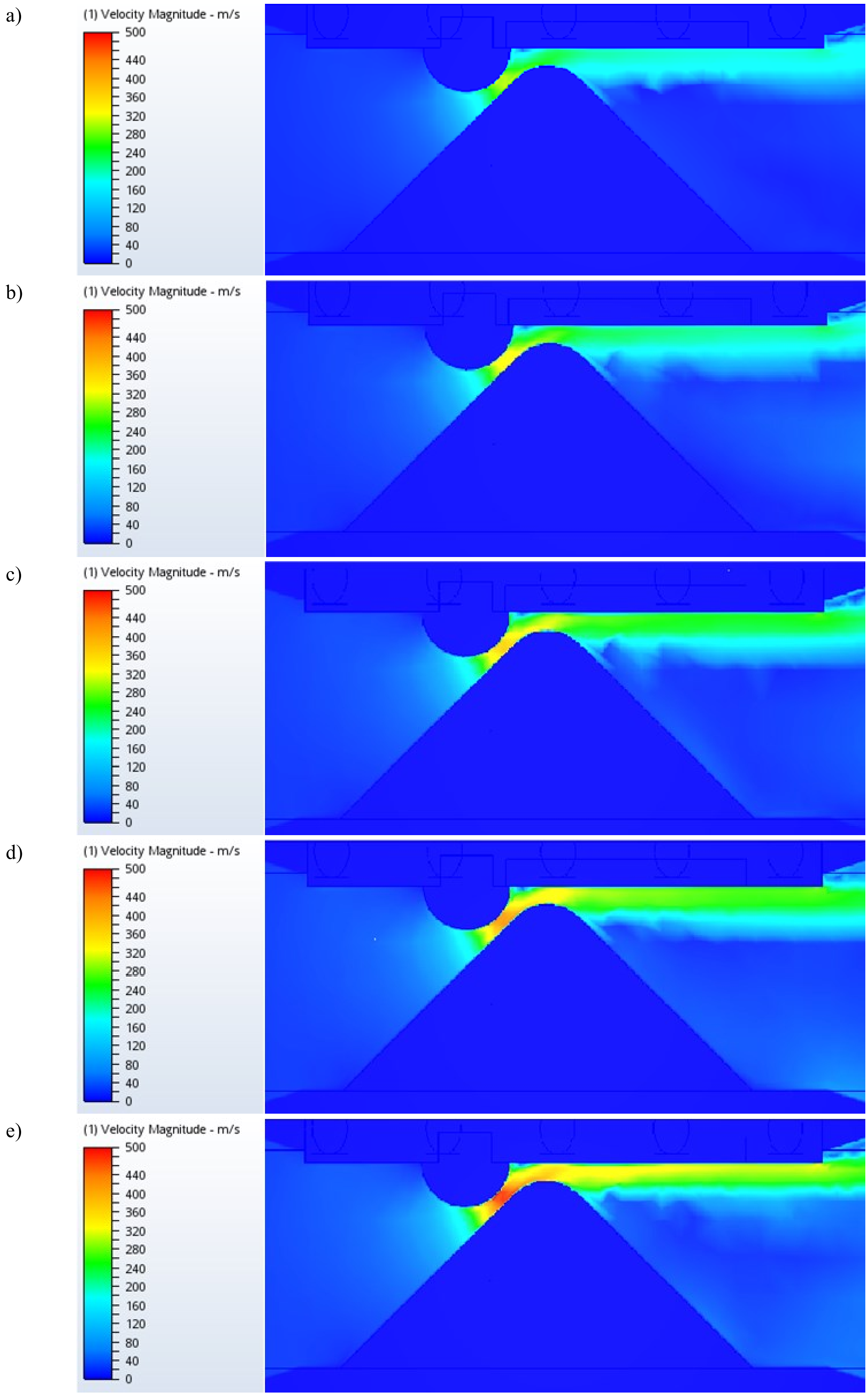

Fig. 6. The results of the velocity in the tunnel for the flow rate in the tunnel inlet: a) 7,74 m/s, b) 8,85 m/s, c) 9,95 m/s, d) 11,06 m/s, e) 12,16 m/s.

DOI: 10.1051/itmconf/20171507011

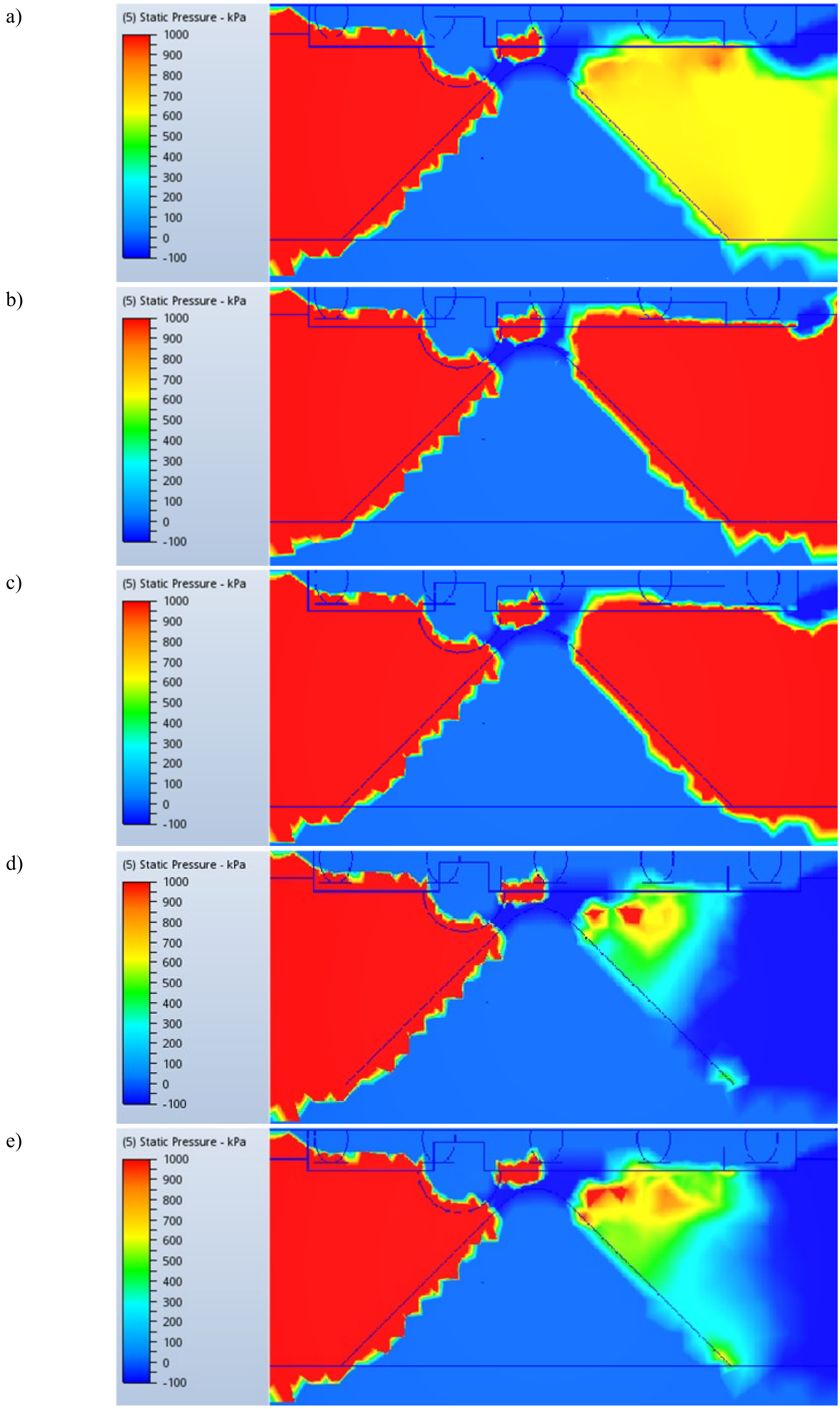

Fig. 7. The results of the pressure in the tunnel for the flow rate in the tunnel inlet: a) 7,74 m/s, b) 8,85 m/s, c) 9,95 m/s, d) 11,06 m/s, e) 12,16 m/s.

The velocity and pressure were analysed in these measurement points. The velocity and pressure were analysed in these measurement points. The results of velocity in the tunnel for all points are summarised as a diagram presented in the Fig. 8.

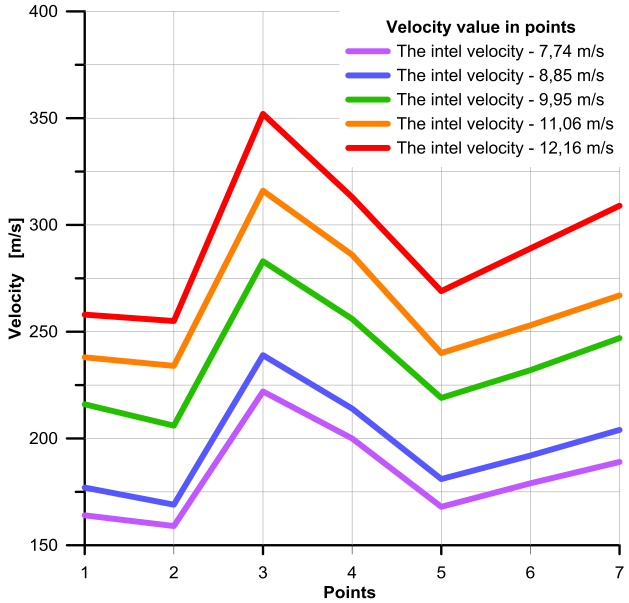

Fig. 8. The results of velocity in the tunnel for measurement points.

The results of pressure in the tunnel are summarised as a diagram presented in the Fig. 9 for all points, and in the Fig. 10 for points from 3 to 7.

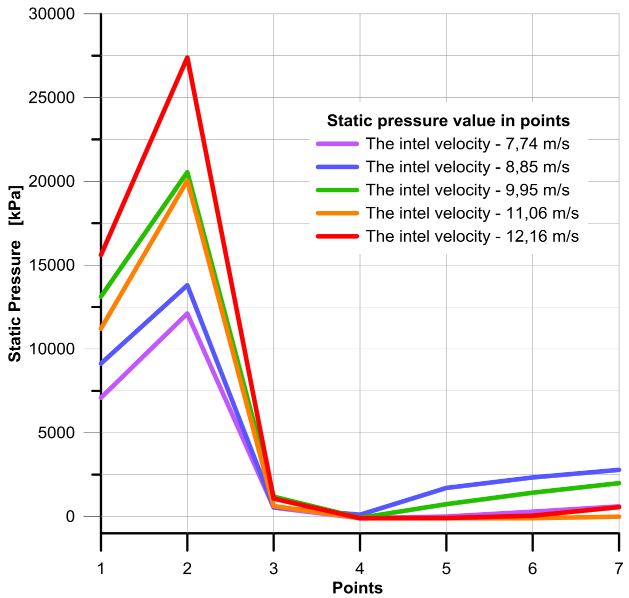

Fig. 9. The results of pressure in the tunnel for all points.

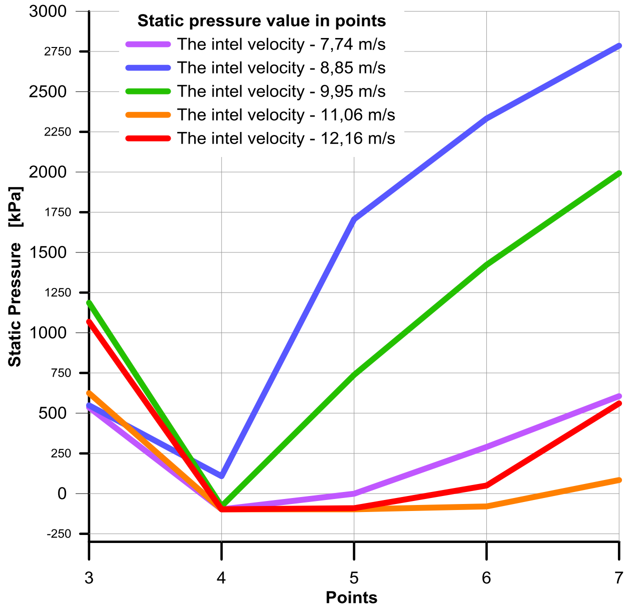

Fig. 10. The results of pressure in the tunnel for points 3 to 7.

4 Conclusions

The analysis of liquid flow with five different flow rates (within the range of 7,74 o 12,16 m/s) through the cavitation tunnel, was carried out in the present work. The conducted numerical calculations allowed determining the flow rate and pressure distribution near the surface of investigated materials. Following conclusions may be drawn from the obtained results:

- the highest values of flow rate (up to 350 m/s) were observed in the third measurement point located in the tunnel, i.e. in its smallest cross-section. The maximum value of flow rate was similar to that achieved in University of Hannover laboratories [7];
- the lowest pressure value between the fourth and fifth point of the measurement, and for all cases, was similar to -98986 Pa;
- the area of lowered pressure increased, raising the initial flow rate to its maximum for 11,06 m/s;
- by analysing all obtained results, it might be stated that the optimal velocity in the tunnel inlet was 11,06 m/s, for which the area of lowered pressure was between the fourth and sixth measurement point.

To sum up, the CFD method has confirmed its usefulness in conducting of simulations in terms of a proper design of machines, devices and laboratory stands. An implementation of this kind of software allows a better understanding of phenomena related to flowing liquids behaviour, as well as to reduce costs associated with experimental verifications. Nevertheless, further experiments regarding various flow geometry will be a valuable supplementary of the presently considered case of cavitation tunnel simulations.

Scientific work funded by the Polish Ministry of Science and Higher Education for statutory activities No. 1/S/IPNT/16.

DOI: 10.1051/itmconf/20171507011

References

1. J. Blazek, *Computational Fluid Dynamics: Principles and Applications*, (Elsevier, 2001)
2. J. Chung, *Computational Fluid Dynamics*, (Cambridge University Press, 2002)
3. T. Cebeci, J.R. Shao, F. Kafyeke, E. Laurendeau, *Computational Fluid Dynamics for Engineers*, (Springer, 2005)
4. A. Sayma, *Computational Fluid Dynamics*, (Ventus, 2009)
5. H. Lomax, T.H. Pulliam, D.W. Zingg, *Fundamentals of Computational Fluid Dynamics* (Springer, 2001)
6. J. D. Anderson Jr. *Computational Fluid Dynamics. The Basics with Applications* (McGraw-Hill, 1995)
7. J. Steller, B.G. Gireń, *International Cavitation Erosion Test. Final Report*, (Institute of Fluid-Flow Machinery Polish Academy of Sciences, 2015)