



Conference Paper

Investigation of Check Valve Aerodynamic Characteristics in Different Operating Modes

D. A. Pakholik, A. V. Sobolev, and A. S. Shelegov

Obninsk Institute for Nuclear Power Engineering of the National Research Nuclear University MEPhI, Studgorodok 1, Obninsk, Kaluga region, 249040, Russia

Abstract

The paper presents a computational study of the NPP ventilation system check valve aerodynamics, namely, a numerical simulation of the air flowing through the open valve with the subsequent determination of the relationship between the reactive torque acting on the valve closure axles and the input air velocity.

This numerical simulation of the air flowing through the check valve was performed using the ANSYS CFX program. In the computation, different operating modes of the check valve were considered when the air flow was passing through it. The valve operating modes were set depending on changes in the input air velocity. As a result of aerodynamic computation, the values of pressure and velocity components were obtained over the entire valve volume.

Reactive forces were calculated in the ANSYS Mechanical program. The reactive forces acting on the valve body form a torque at the gate axles. When adjusting the check valve to the actual flow rate, it is necessary to know this torque value and compensate for it. As a result of a series of computations of reactive forces, a relationship was found between the torque value of the valve's working element axles and the input air velocity.

Keywords: investigation of aerodynamic characteristics, check valve, reactive forces, operating characteristic.

Corresponding Author:

D. A. Pakholik
pakholikda@mail.ru

Received: 23 December 2017

Accepted: 15 January 2018

Published: 21 February 2018

Publishing services provided by
Knowledge E

© D. A. Pakholik et al. This article is distributed under the terms of the [Creative Commons Attribution License](#), which permits unrestricted use and redistribution provided that the original author and source are credited.

Selection and Peer-review under the responsibility of the AtomFuture Conference Committee.

1. INTRODUCTION

One of the most important issues to be solved at each enterprise in the nuclear industry is safety, a constituent part of which is radiation safety. Radiation safety issues are addressed through the integrated protection of biological objects and humans to create safe environmental conditions, including land, water and air. Ventilation, as one of the ways to ensure radiation safety, serves to maintain the cleanliness of the air environment and prevent its contamination, both inside the NPP premises and, indirectly,

 OPEN ACCESS

outside the NPP. Current requirements of norms and rules for the use of nuclear power [1, 2] determine the necessity for ensuring high quality of NPP ventilation.

One of the goals of ventilation at NPPs is to ensure a certain air flow direction by using various devices, both at its inflow and outflow [3]. These devices include check valves, which admit air in one direction and prevent it from moving in the opposite one, while acting automatically [4-6]. The check valves maintain the operability of treating facilities and fans, which are used in the NPP ventilation system and, therefore, carry a large preventive load as means of environmental protection.

This paper presents an investigation of the aerodynamic characteristics of the NPP ventilation check valves. Studies of this kind are currently relevant, because the existing experimental data do not provide necessary reliability as to the level of aerodynamic resistance and loading of check valves. In addition, valve testing is an extremely costly procedure, whereas, using computational methods and reducing the scope of tests, it becomes possible to lower the product cost, thereby increasing its competitiveness.

2. CHECK VALVE AERODYNAMICS COMPUTATION

The computational domain is the flow path of a check valve model shown in Fig. 1. This valve contains a body (pipe diameter: DN 1250) with a closure installed in it. The valve closure rotation axles are installed out of its alignment and equipped with a lever with a movable load.

Let us consider the operating principle of a check valve of the presented design. When there is no medium flowing through the valve, the gate affected by the load is in closed position. When a flow occurs, the gate, under the action of its energy, opens the passage through the valve. It is obvious that for the medium flow to change its direction to the opposite one, it should be stopped. At this moment, the flow rate becomes zero, the gate returns to its initial, closed position, and the pressure from the back side presses it, preventing the medium from flowing back. Thus, the check valve actuates under the action of the medium itself and is fully automatic.

2.1. Numerical simulation of the air flowing through the check valve

The numerical simulation of the air flowing through the open check valve was performed using the ANSYS CFX program [7, 8]. This program realizes a finite-volume

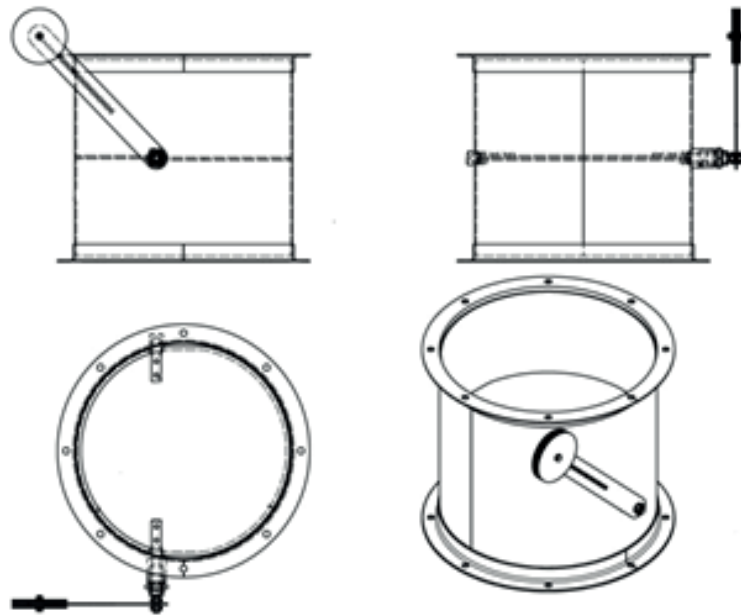


Figure 1: A check valve model.

method for solving equations such as the continuity equation, the energy conservation equation, and the momentum equation.

Structurally, ANSYS CFX includes the following modules: CFX-Pre (preprocessor), CFX-Solver (solver) and CFX-Post (postprocessor) [7]. In addition, when setting the task, the following programs were used: ANSYS DesignModeler [9] – for working with the geometry, and ANSYS Meshing [10] – for working with the computation mesh.

2.1.1. Check valve computation model construction

Constructing a computation model is an important process in solving aerodynamic problems on a computer. The quality and speed of solving problems directly depends on this process.

To create this computation model, a three-dimensional check valve model was developed. The model was developed using the CAD system that has a function of importing data into ANSYS CFX. The CAD system is an automated design tool: in this context, the term means a tool designed to automate a two-dimensional or three-dimensional geometric design.

In order to reduce the computation mesh dimension and computation time, the following simplifications of the three-dimensional model were adopted:

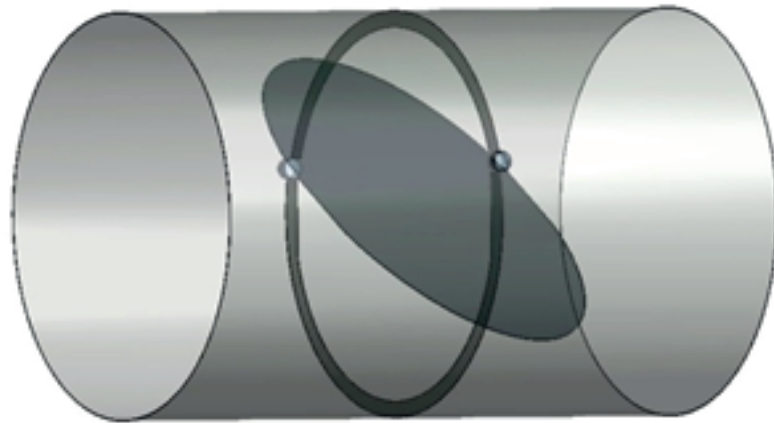


Figure 2: Computation model.

- The check valve body is made in the form of a single pipe; the flange connection of the pipeline with the valve body is not taken into account;
- The gate with fastenings is made as a single solid-state component.

To create the computation model, the geometric parameters of the check valve developed in the CAD system were imported into the ANSYS DesignModeler program, where a solid valve model was adapted to generate the mesh. The model adaptation includes “filling” the valve flow path with air and identifying contact regions.

A general view of the computation model created in the ANSYS DesignModeler program is presented in Fig. 2.

2.1.2. Computation mesh construction

Constructing a computation mesh is a process of dividing the computational domain into a set of separate cells. Mesh cells, as a rule, are tetrahedra, hexahedra, prisms or pyramids. The main requirement for the computational mesh is that it must be sufficiently dense to allow the physical effects occurring within the computational domain. In order to achieve uniform computational accuracy, the mesh nodes should be located more densely at places of large gradients of flow parameters, in particular, near the walls [10, 11].

The mesh for computing the check valve aerodynamics was generated in the ANSYS Meshing program. The computation model mesh was made using tetrahedra and is shown in Fig. 3.

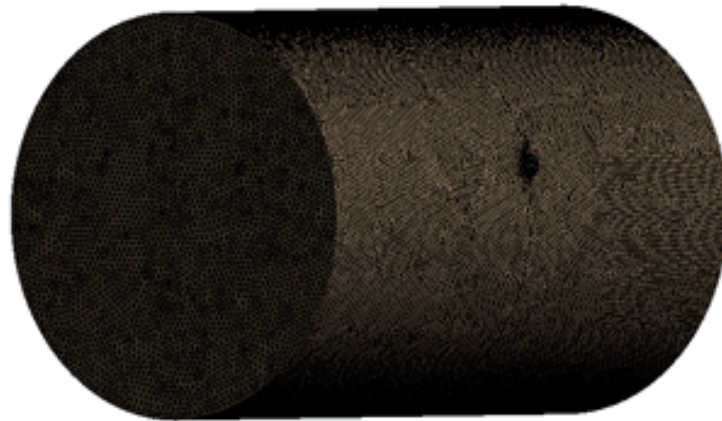


Figure 3: Computation mesh.

2.1.3. Boundary conditions

For the numerical simulation, it is required to specify the values of dependent variables or their normal gradients at the boundaries of the computational domain. After the mesh is generated, the computation model is imported into the ANSYS CFX-Pre application, where a physical model is defined, based on which the computation will be performed, including the search for the main parameters and characteristics.

As long as there are always several types of boundary conditions (BC) in any problem, the question arises as to their optimal combination and even the correctness of jointly using their certain types. The most stable combination of boundary conditions is setting the velocity or mass flow at the inlet and static pressure at the outlet of the computational domain [12, 13]. In this case, the total inlet pressure is determined by computations.

In view of the above, the following initial data were used to specify the boundary conditions:

- Inlet speed: 5, 10, 20, 30, 39, 40, 50, 60, 70, 80 m/s;
- Air temperature: 25 °C;
- Average static outlet pressure: 1 atm;
- The computational model takes into account all the basic properties of air, depending on temperature and pressure;
- The turbulence model $k-\varepsilon$ was used.

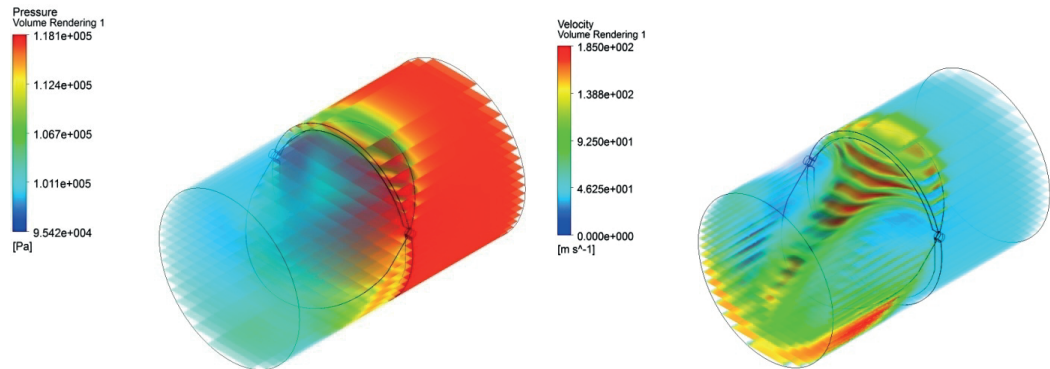


Figure 4: Pressure and flow rate volume distribution (inlet velocity: 39 m/s).

2.2. Reactive torque computation in different check valve operating modes

The initial data for computing the reactive torque in different valve operating modes are the computation results obtained in ANSYS CFX, namely, the pressure differences at the check valve gate for different flow rates. ANSYS Mechanical was used for the computations [14-16].

The check valve gate anchorage (i.e., the prohibition of all 6 degrees of freedom) was carried out in the region where the axles were attached to the body. The check valve piping connections were not taken into account. The material of the welds was considered identical to the valve basic material without loss of strength.

3. RESULTS OF THE COMPUTATIONAL RESEARCH

3.1. Computation results of the air flowing through the check valve

As a result of computations of the air flowing through the check valve, the velocity and pressure components were obtained. Figures 4–6 show some computation results in the form of distributions of the air parameter (pressure and velocity) fields along the flow path and streamlines.

The resulting parameter distributions that determine the pattern of the air flowing through the check valve are quite predictable. In all input velocity variations, the flow is accelerated and the air pressure drops after the gate, where eddying occurs.

Take a note of the air flow at the top of the check valve. Immediately after the gate, the flow separates to form a developed eddying zone. The rest of the flow, passing

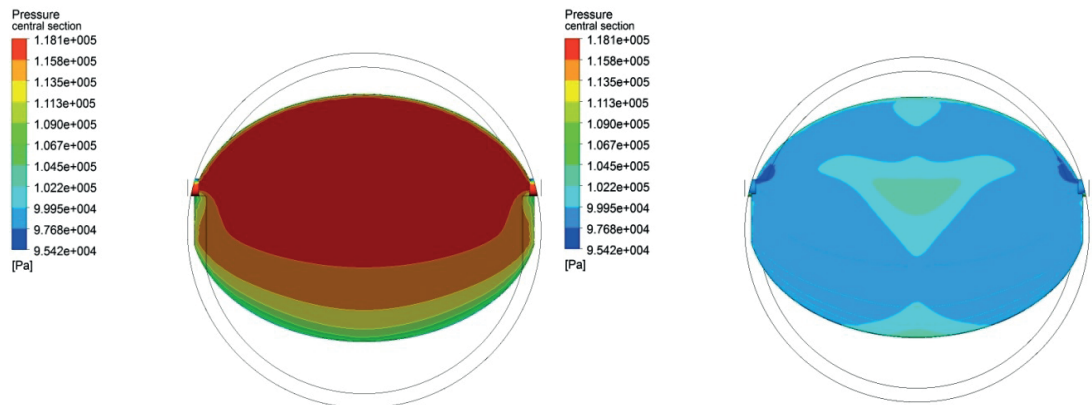


Figure 5: Air pressure distribution through the gate from the flow inlet (left) and outlet (right) (inlet velocity: 39 m/s).

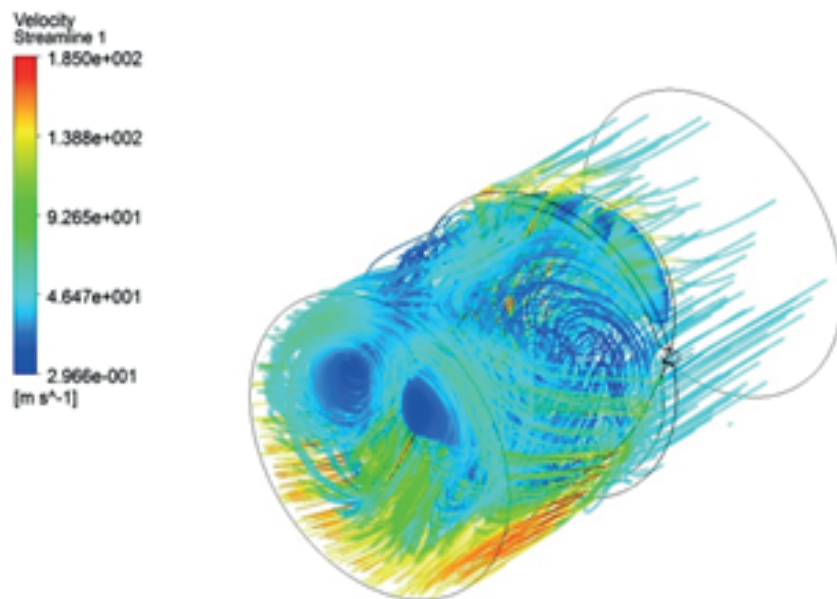


Figure 6: Air flow streamlines in the check valve (inlet velocity: 39 m/s).

through the gate from below, moves without separation, bypassing the eddying area. The pressure field after the gate is very inhomogeneous, which is typical for all valves with a single medium inlet. The velocity field turns out to be equally inhomogeneous.

The air pressure drop trend while passing through the check valve can be described by a dependence as $\Delta P = f(V)$. Figure 7 shows the relationship between the pressure drop and the inlet air velocity obtained as a result of the computation.

The obtained pressure drop data are necessary for designing a ventilation system using the considered check valve.

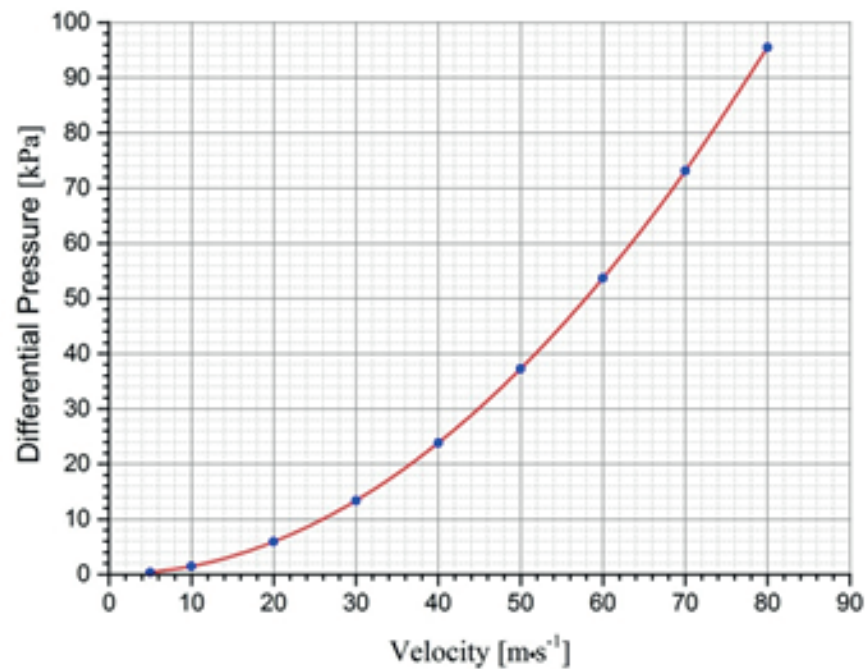


Figure 7: Differential pressure curve in the check valve depending on the inlet air velocity.

3.2. Reactive torque computation results

The results of the numerical simulation of the air flowing through the check valve make it possible to estimate the forces acting on individual components of the valve structure. The reactive forces acting on the valve body form a torque at the gate axles. To adjust the check valve to the actual flow rate, it is necessary to know this torque value and compensate for it. The speed-torque curve for the valve body axles is shown in Fig. 8. It can be seen from the graph that at a flow velocity of ≈ 39 m/s the reactive torque is close to zero but at higher velocities it changes its direction. The obtained value makes it possible to determine the check valve operating range. Thus, at a flow rate of less than 39 m/s, the check valve actuates and interlocks the flow. The obtained characteristic can be used when setting the check valve on site. In fact, this is the check valve operating characteristic.

4. CONCLUSION

Computational research methods make it possible to obtain a detailed picture of the level of air resistance and power loads arising in various pieces of equipment. In particular, these results are necessary for designing check valves. This approach significantly

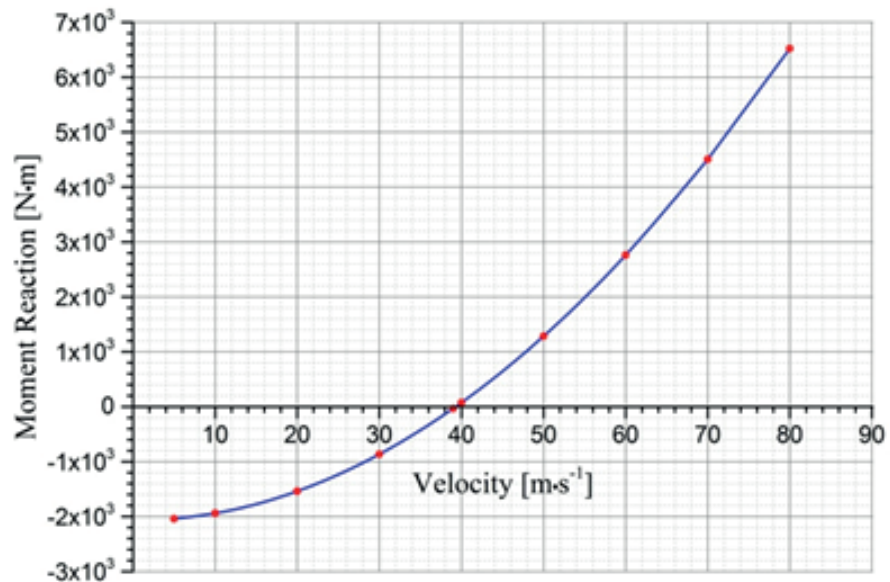


Figure 8: Relationship between the reacting torque and inlet air velocity in the check valve.

reduces the volume of experimental studies and, as a consequence, reduces the product cost.

The performed computational study of the check valve aerodynamic characteristics made it possible to obtain pressure and air velocity fields from the NPP ventilation system as well as reactive forces acting on the valve closure body. In addition, graphs of relationships between the pressure drop and torque on the valve axles and the flow velocity were plotted. The results obtained clearly show the check valve operation and can be used for adjusting the fittings to the required parameters.

References

- [1] NP-010-16: Federal Standards and Rules on the Use of Atomic Energy "Rules for Design and Operation of Confining Safety Systems of Nuclear Plants", 2016.
- [2] NP 036-05: "Rules for Design and Operation of Safety-Important Ventilation Systems of Nuclear Plants (No. 2)", 2006.
- [3] Margulov T.Kh. Nuclear power plants: Textbook for high schools. 4nd edition. Moscow, Vysshaya Shkola Publ., 1984. 304 p.
- [4] Gurevich D.F. Pipeline fittings. Reference book. 2nd edition. Leningrad, Mashinostroenie Publ., Leningrad Branch, 1981. 368 p.
- [5] Gurevich D.F. Computation and design of pipeline fittings: Computation of pipeline fittings. 5th edition. Moscow, LKI Publ., 2008. 480 p.

- [6] Bashta T.M. Hydraulic drive and hydropneumoautomatics. Moscow, Mashinostroyeniye Publ., 1972. 320 p.
- [7] ANSYS CFX Reference Guide, ANSYS, Inc., 2013. 386 p.
- [8] Kaplun A.B., Morozov E.M., Olferyeva M.A. ANSYS in the hands of an engineer: Practical guide. Moscow, Knizhnyidom "Librokom" Publ., 2015. 272 p.
- [9] ANSYS DesignModeler User's Guide, ANSYS, Inc., 2012. 548 p.
- [10] ANSYS Meshing, User's Guide, ANSYS, Inc., 2010. 350 p.
- [11] Workbench User's Guide, ANSYS, Inc., 2013. 286 p.
- [12] ANSYS. One-stop ANSYS product support center in Russia and CIS countries [Electronic resource]. URL: <http://cae-expert.ru>(Reference date: 11.05.2016).
- [13] ANSYS [Electronic resource] // URL: <http://www.ansys.com>(Reference date: 18.03.2016).
- [14] ANSYS Mechanical User's Guide, ANSYS, Inc., 2013. 1832 p.
- [15] Bruyaka V.A. et al. Engineering analysis in AnsysWorkbench. Manual. Part 1. Samara: SSTU publ., 2010. 271 p.
- [16] Bruyaka V.A. et al. Engineering analysis in AnsysWorkbench. Manual. Part 2. Samara: SSTU publ., 2013. 149 p.