Modern Applied Science; Vol. 10, No. 2; 2016
ISSN 1913-1844 E-ISSN 1913-1852
Published by Canadian Center of Science and Education

New Flow Conditioners as a Means of Enhancing the Reliability and Efficiency of Power Equipment

Arkadiy Efimovich Zaryankin[1], Nikolay Dmitrievich Rogalev[1], Andrey Nikolaevich Rogalev[1], Anna Aleksandrovna Kocherova[1] & Galina Nikolaevna Kurdiukova[1]

[1] Moscow Power Engineering Institute, National Research University, Moscow, Russian Federation

Correspondence: Anna Aleksandrovna Kocherova, Moscow Power Engineering Institute, National Research University, 14, Krasnokazarmennaya street, Moscow, Russian Federation. E-mail: kocherovaaa@yandex.ru

Received: August 12, 2015 Accepted: November 10, 2015 Online Published: January 7, 2016
doi:10.5539/mas.v10n2p172 URL: http://dx.doi.org/10.5539/mas.v10n2p172

Abstract

This paper researches the nature and the causes of the arising of the dynamic loads, induced by the pressurized water flow in pipes and affecting their walls, considering the case of flow in a curved component of a steam supply system of a steam turbine as an example, and examines the flow conditioners that are currently utilized to reduce the loads. Having analyzed their faults, we suggest new configurations of flow conditioners. The new configurations reduce the cross-section of a pipeline half as less and, consequently, they are expected to provide a significantly lower hydraulic resistance. The efficiency of the new configurations was assessed by means of the CFD methods and compared against the widely used current models. As the results of the comparison show, the new configurations demonstrate a comparable degree of the reduction of the dynamic loads. However, their hydraulic resistance is five times lower. Their features also allow to expect better durability characteristics.

Keywords: a flow conditioner, vibration, velocity field, roughness, an eddy, vortex formation

1. Introduction

1.1 How the Dynamic Loads come to be

The analysis of accidents at power plants and technologically related facilities shows that the most common cause of damage to the equipment, for example, to thermal hydraulic installations, safety valves and pipelines, is the dynamic loads induced by the flow of the working medium and affecting their parts. The most stressed parts are those that undergo a polar change in the direction of the flow of the working medium. This happens due to centrifugal forces that throw liquid particles to the outer (concave) walls of a curved channel, which results in a lateral pressure gradient (Figure 1) (Deitch, 1961; Khazaei et al., 2010).

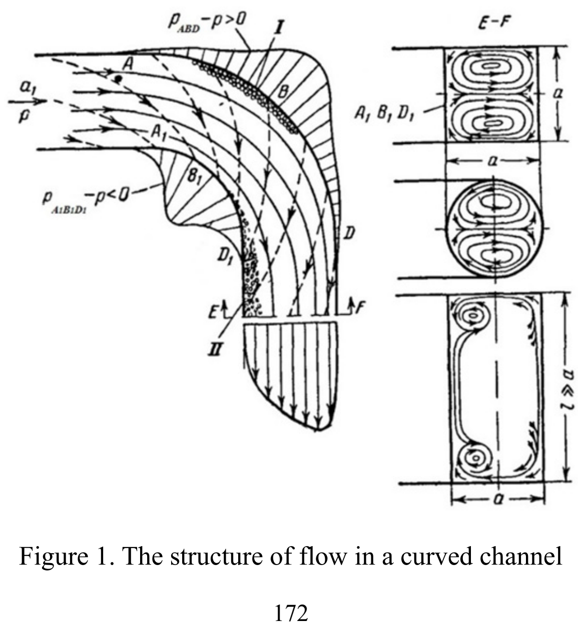


Figure 1. The structure of flow in a curved channel

Hence, this creates additional conditions for shedding separated eddies, which ultimately leads to the emergence of the non-laminar profile of the flow velocity. The roughness and non-stationary character of the velocity field, in turn, mean the equally non-uniform character of pressure within the flow (Karavosov & Prozorov, 2014). Therefore, to achieve a greater degree of reliability of power equipment, we need to analyze in detail the character of the dynamic loads, as well as the structure of the flow that induces them, and develop an efficient way of their prevention.

A frequently met case of flow in a curved channel takes place in the steam input system of steam turbines. The forces of pressure, being time-dependent, affect the pipe's walls and cause vibrations of a wide spectrum of frequencies. The results of an experimental research on the pressure pulsations in the steam input parts of a steam turbine (Figure 2), whose initial parameters of the nominal mode are 24 MPa, 540°C, showed that the amplitudes of the low-frequency pulsations (3, 3; 7, 5; 10, 5 Hz) amounted, in average, 2, 5 MPa, and, in some cases, even 4÷5 MPa; the modes of the tests varied. The amplitudes of the pressure pulsations at the range of 400÷450 Hz amounted 2.3÷2.5 MPa, and at higher frequencies (for instance, up to 1150 Hz) they do not exceed 1.0÷1.2 MPa. However, in the latter case the energy of the pulsations can be significantly higher, as a result of a high frequency (Kostiuk et al., 2000).

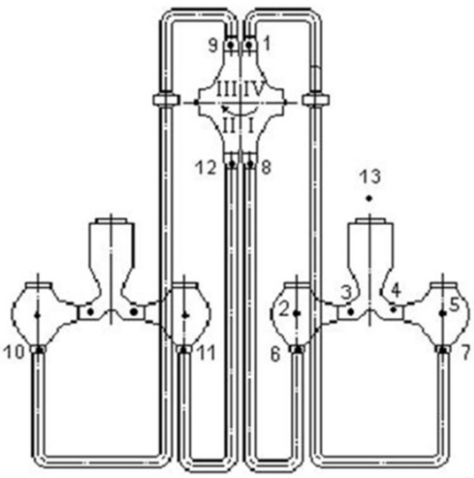


Figure 2. The steam input system of a steam turbine, the sensors' location is shown

Obviously, such pressure pulsations have a most negative effect on the equipment's reliability. It is known that the frequency of the pulsations is directly dependent on the size of eddy structures: the larger the swirls are, the lower the frequency of pulsing is and the higher is its energy potential. As the swirls grow less in size, their velocity within the flow grows equal with the velocity of the basic flow, the frequency of the pulsations they generate is increasing, and the general energy of the swirl motion decreases sharply. Therefore, the main way of reducing the dynamic loads on the channel walls is to split the large eddy cores into smaller structures and to spread them homogeneously in the channel's cross-section (Grigoriev, 2014).

This principle of counteracting the pressure pulsations is implemented in the so-called flow conditioners, or stabilizers (aerodynamic filters, or swirl dampers), which most often look like perforated discs of various configurations, installed into pipelines (Figure 3).

1.2 The Dilemma of a Flow Conditioner

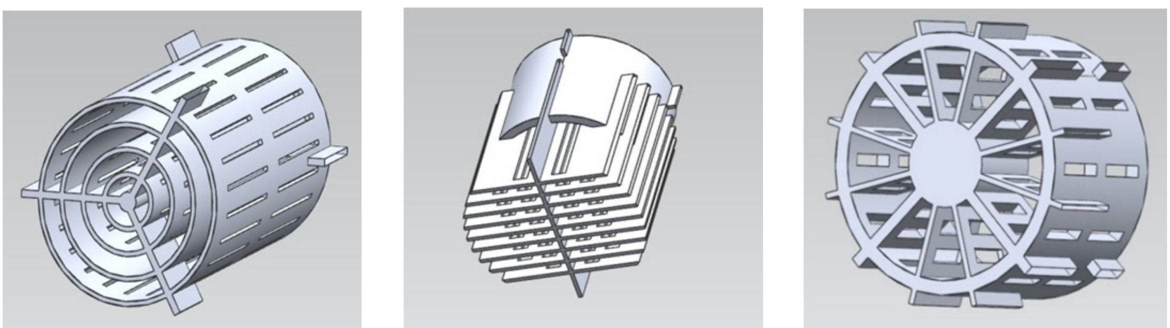
The process of splitting the eddies shedded consumes energy, but this is compensated by a decrease in the hydraulic resistance in the further sections of a pipeline, along with the further decrease of the eddy structures in size and energy; and they are flowed around by the laminar part of the flow like rigid bodies. However, it is obvious that the installation of a flow conditioner will inevitably lead to a rise in the system's hydraulic resistance, since this reduces the cross section of the flow. The pressure losses tend to be very substantial. In particular, the hydraulic resistance coefficient of a disk flow conditioner, similar to that shown in Figure 3, is equal to $\zeta = 10$, and it is noted in (Deitch, 1974; Zaryankin et al., 2014) that the degree of flow smoothing, provided by disc filters, is proportional to the resistance coefficient ζ and, therefore, trying to reduce the resistance by simply increasing the number of perforation holes, we only achieve a worse efficiency of the filter in question. As an example of that can serve the case of installation a conditioner in the shape of a perforated cylindrical screen into the compartment of the regulating section of the turbine K-200-130 LMZ. This measure led to a substantial decrease in the level of vibrations of the turbine unit, but the hydraulic resistance of the conditioner turned out to be so high that the eventual economic effect was equal to none (Zaripov & Mikheev, 2014; Huang et al., 2012). Hence, the current challenge is designing such flow conditioners that would not only effectively decrease the dynamic loads on the parts of the equipment, but also have a low hydraulic resistance.

a b c

Figure 3. The currently used flow conditioners: a–a perforated disc flow conditioner; b a slot-type flow conditioner; c–a duplex slot-type flow conditioner

In order to achieve that, we need, first of all, to provide a greater area of the flow cross section of the device. The most effective solution would be to reduce the surface of the conditioner that faces the flow, because the swirls can as well be split by means of interaction of the medium with thin surfaces, positioned at narrow intervals along the direction of the flow. The perforation of the conditioner's surfaces will also contribute to decreasing the hydraulic pressure, as it will allow the liquid to easily move under the influence of the lateral pressure gradient, that arises when the flow turns, and, secondly, this will lessen the difference in the pressures, that influence the parts of the device in the frontal (cross section) plane, which will add to its reliability.

Taking all these considerations into account, the researchers at the Moscow Power Engineering Institute (MPEI) proposed some new variants of the flow conditioner configuration (Figure 4).



a b c

Figure 4. The suggested configurations of flow conditioners: a–a linear circumferential flow conditioner; b–a plate-type flow conditioner; c–a sectoral flow conditioner

These devices are presumed to smooth the flow velocities profile with no less efficiency than their analogues, described above, but, at the same time, to be able to boast of a significantly lower hydraulic resistance. An optimal way to check this hypothesis at the stage of preliminary design will be to utilize numerical methods, namely, the CFD-modeling.

2. The Methods of Simulation and Assessment of the Flow Parameters in the Channels of the Flow Conditioners

2.1 The Computation Models for the CFD Flow Simulation

The efficiency of the newly devised flow conditioners should be estimated against the results of the previous types, therefore, in order to justify the implementation of the new configurations, we need to devise 3 groups of models:

1) The simulation of the flow in a curved pipeline without a flow conditioner (the initial / baseline case);

2) The simulation of the flow in a curved pipeline after the installation of an existing flow conditioner;

3) The simulation of the flow in a curved pipeline after the installation of a new configuration of a flow conditioner.

The research by means of the numeric modeling methods is conducted according to the following procedure:

- creating a geometric three-dimensional computation model of the inside of the equipment in question;
- dividing the computed volume into a finite number of elementary units (generating a computation mesh);
- carrying out three-dimensional CFD calculations:
    - setting the boundary conditions;
    - configuring the solver;
    - setting the iterative character of the problem solving process.

As the computational environment for the processing of geometric models we chose the software pack "SolidWorks". The SFD-simulation requires to have a three-dimensional model of the inside of the investigated equipment. The computation model for researching the initial variant of the flow is represented in Figure 5, for researching the flow after the installation of a conditioner–in Figure 6.

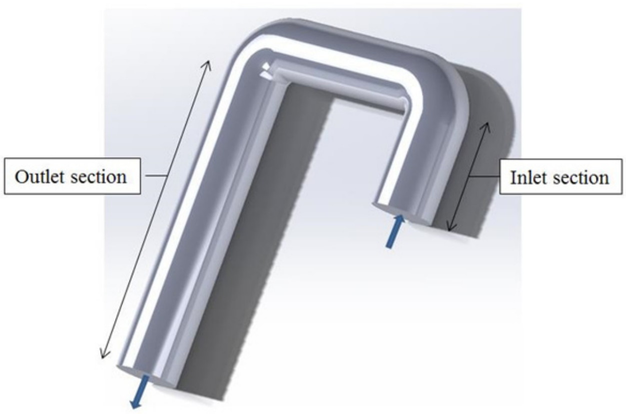


Figure 5. The 3D-model of a curved component of the steam supply system of a steam turbine, D_{in} = 100 mm

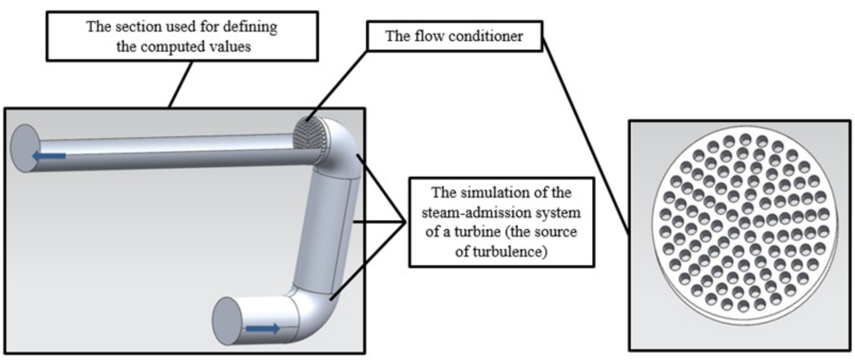


Figure 6. An example of the 3D-model of a curved component of the steam supply system of a steam turbine with a flow conditioner installed, D_{in} = 100 mm

2.2 The Method of Flow Simulation

The 3D model is imported into the meshing generator “Ansys ICEM”. Mesh (grid) generation is a discretization of a computation (spatial) domain into a piecewise structured array of cells. The mesh cells are polyhedra, usually tetrahedra, hexahedra, prisms or pyramids (Figure 7). The edges of the cells are the computation mesh’s lines, and the points on the edges or in the cells’ center–are the mesh’s nodes. It is in the nodes of the computational mesh that the sought flow parameters are defined, as a result of the numerical solution of the equations of the mathematical model.

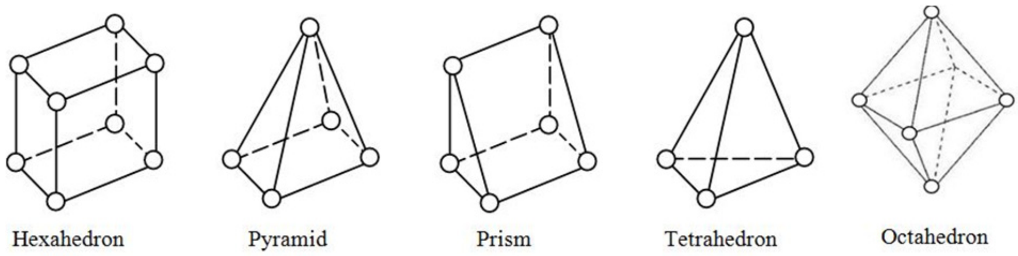
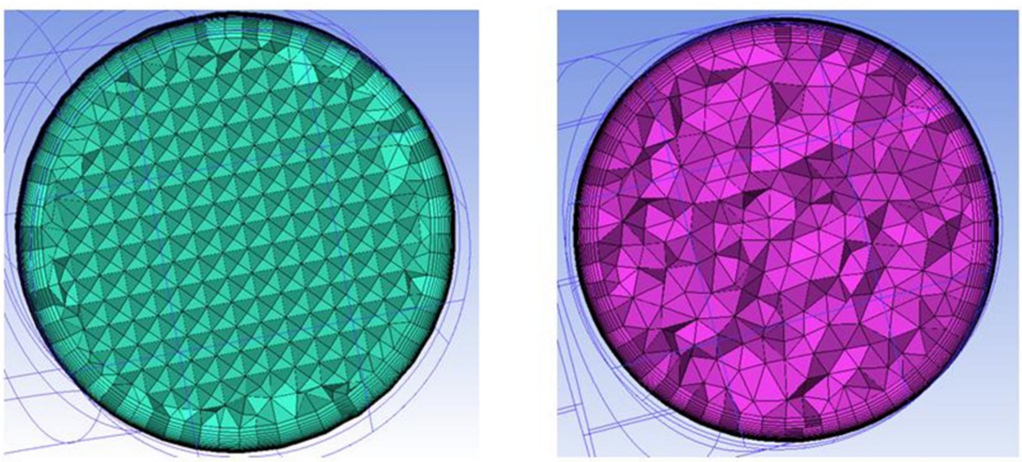


Figure 7. The standard shapes of mesh cells

The computational grid should be dense enough, in order to render the physical processes within the volume of calculation with a sufficient resolution. To achieve a homogeneous accuracy of the calculation, the grid’s nodes should be located more densely in the areas of rapid change of the flow parameters, in particular, near the walls.

First, we create a three-dimensional grid by the Robust (Octree) meshing method, when the space consists of octahedra elements. Thus obtained mesh is then re-generated, and a new mesh is created by the Quick (Delaunay) method, which uses the octahedra mesh as an initial approximation. The latter method has a more complicated algorithm, but is better suited for the tasks of processing complex geometry and allows for smoother transitions between the sizes of the elements. However, for simple tasks, like the analyzed case of flow in a curved pipe, the octahedra mesh will suffice (Kim et al., 2014; Lyubimov & Shabarova, 2011; Pulat & Ersan, 2015).

Finally, at this stage, in order to simulate the flow in the boundary region most adequately, the prism-shaped elements are build over the wall and in the near-the-wall region, that are piled in a several layers and grow in size towards the core of the flow (Figure 8). The parameters of the prismatic layer are selected to ensure the value of the parameter $y^+ < 1$. Eventually, the computational grid's size, as a rule, amounts approximately 4-5 million nodes (Arshad et al., 2010; Martins et al., 2014).



a b

Figure 8. The three-dimensional mesh with the prismatic layer: a–by the Robust / Octree method, b–by the Quick (Delauney) method

The generated mesh is imported into the program "Ansys Fluent", where the methods of solving a problem in fluid dynamics are selected. The system of equations that describe the examined process includes:

- the continuity equation;
- the momentum-conservation equation (RANS–Reynolds-averaged Navier-Stokes);
- the energy-conservation equation;
- the turbulence model equation.

Various models of turbulence require different types of meshes. The above type of mesh is suitable for a *k-ε* turbulence model that has proved to be the most adequate for the solution of our problem (Laribi et al., 2012). Therefore, the following settings are used for solving the equations (Li et al., 2010):

- the settings of the solver: RANS pressure based solver;
- the turbulence model: releasable *k-ε*, enhanced wall treatment;
- the second order of the accuracy of discretization.

The procedure of solving the system of equations that describe the fluid dynamics processes also requires setting the boundary conditions. At the outlet, we set the constant pressure, equal to the atmospheric one at the inlet we set the boundary conditions of fixed flow rate. That means, that when solving the problem, the full pressure at the inlet is being changed until we obtain the required flow rate. The air temperature at the inlet is 20°C, the temperature of the exhausts at the outlet is also 20°C. The walls are viscous, adiabatic. As the working medium, the air is assumed, whose physical properties depend on the temperature. The air flow rate is adjusted so that the average velocity of the flow would be 60 m/s, which is typical for power equipment.

The solutions of fluid dynamics tasks, obtained with this methodology, agree with the empirical data with sufficient accuracy, as regards both the structure of the flow and the quantitative parameters (for instance, the error for hydraulic resistances is less than 1%). So, despite the fact that the CFD-modeling cannot replace experimental research, it is a reliable instrument for researching the fluid dynamics processes that have place in practice.

2.3 The Method of Assessing the Flow Conditioners' Efficiency

The quantitative estimation of the effect of the suggested swirl dampers on the flow in a curved channel is carried out by means of two parameters:

- by the hydraulic resistance coefficient:

$$\zeta = \frac{P_1 - P_2}{\frac{\rho c^2}{2}} = \frac{P_1 - B}{\frac{\rho c^2}{2}} \tag{1}$$

- by the coefficient of roughness (non-laminarity) (Tsinoglou et al., 2004):

$$\omega = \frac{\Sigma(c_i - \bar{c})}{\bar{c}}, \tag{2}$$

where c_i stands for the velocity in the i^{th} point of the profile, $i = 1 \dots n$; $\bar{c}$ is the average velocity in the profile.

The basic configurational characteristics of the flow conditioners are:

- the coefficient of the surface perforation, equal to the ratio of the area of the holes to the area of the same surface without perforation:

$$f = \frac{F_{perf}}{F_{\Sigma}} \tag{3}$$

- the flow cross-section coefficient, equal to the ratio of the area of the cross-section before the installation of the conditioner to the same area after the installation:

$$s = \frac{S_{stab}}{S_{baseline}} \tag{4}$$

The latter coefficient is introduced to characterize the newly devised conditioners, whose basic working surface is oriented along the direction of the flow in a pipeline, which means that the coefficient of perforation would not give the information about the cross-section.

3. The Results of the Numerical Simulation

The results of the simulation of the flow in a curved channel that imitates a part of the steam-supply system of a turbine unit give evidence of that large eddies are shed in the channel's curves, and they create perturbances along considerable lengths of a pipeline (Figure 9) (Bandyopadhyay et al., 2010).

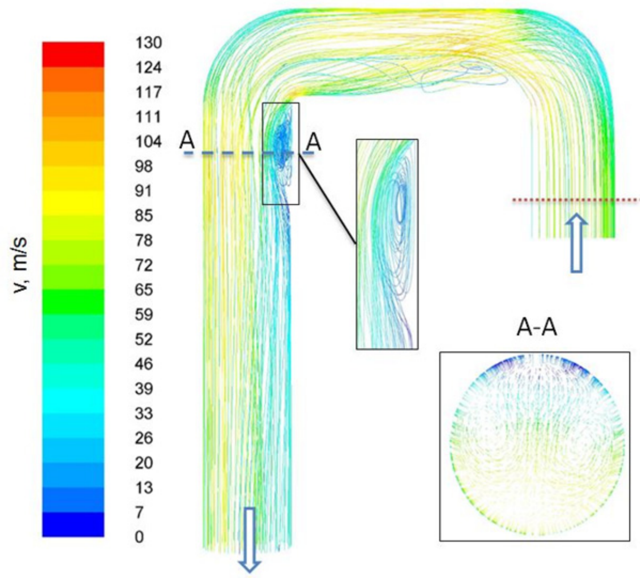
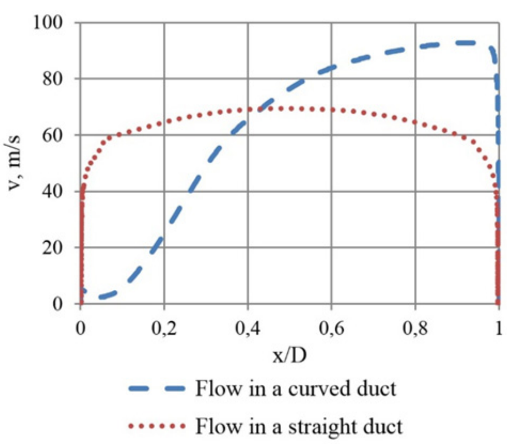


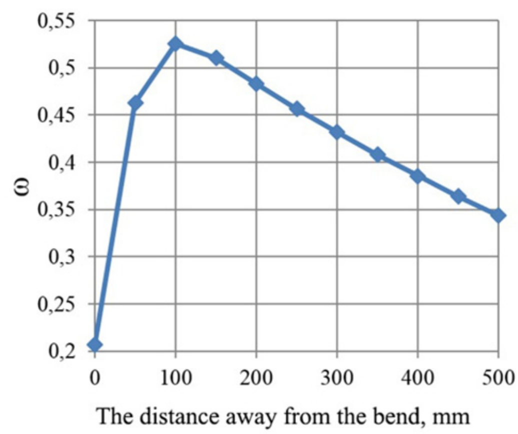
Figure 9. The results of the flow simulation in a curved channel, imitating a part of the steam supply system of a turbine unit

The velocity profile of the flow, as taken right after its turn (Figure 10, a), also gives evidence of the presence of the separated currents and of the pressures field being very uneven. For comparison, we give the velocity profile of a laminar flow that takes place in a straight pipe with the same mass flow rate.

The type of the dependency of the roughness coefficient on the distance to the curved element (Figure 10, b) also testifies the presence of a swirling current that arises right after the flow's turn. Its effect is still significant even at the distance of 500 mm–the roughness coefficient, calculated by the methods described above, in a straight pipe would be equal to approximately 0.05, while, at the distance of five diameters to the source of perturbance, it is equal to 0.34.



a



b

Figure 10. The characteristics of the flow in a curved channel: a–the velocity profile of the flow after the bend; b–the distribution of the index of roughness along the channel

Figure 11 illustrates the flow in a curved channel with one of the most widely used type of flow conditioners installed – a disc conditioner. Prior to the installation of the stabilizer, as we can see from Figure 10, the eddy occupied approximately 40% of the pipeline's cross-section, and the perturbance caused by it persisted at a distance of no less than five diameters. Now the eddy structures have grown significantly smaller in size, and, as we can judge from the diagram for the roughness coefficient (Figure 12), they fade practically immediately, at the distance of half a diameter.

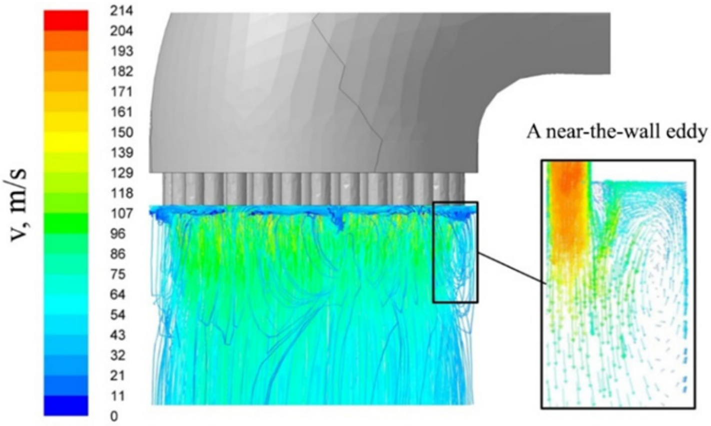


Figure 11. The results of the flow simulation for a perforated disk conditioner

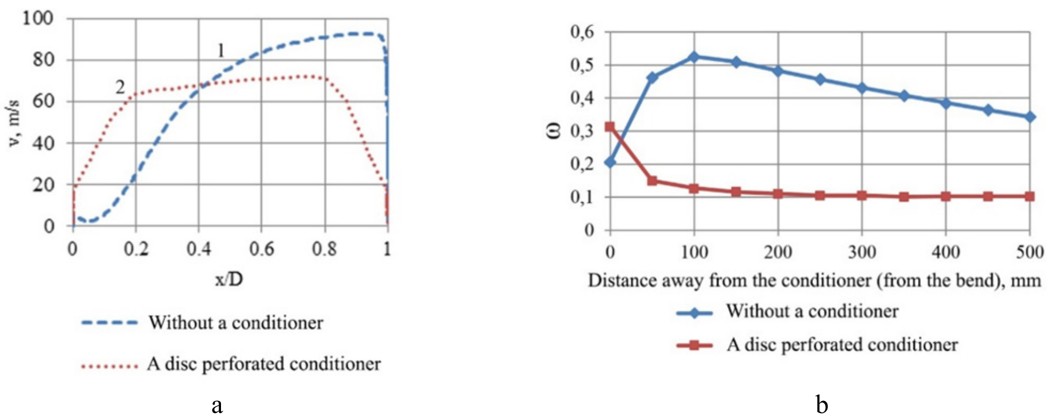


Figure 12. The characteristics of the flow in a curved channel with a perforated disk conditioner: a–the velocity profile of flow after the bend; b–the distribution of the index of roughness along the channel

Still, as expected, the coefficient of hydraulic resistance of the device turns out to be high – it amounts 10.8.

The results of the calculations for the flow in a curved channel when flowing around one of the newly devised conditioners – the plate-type one – are given in Figure 13. Additionally, the plate-type conditioner was installed in the pipeline in such a way that its plates were in the plane of the turn, since that allows to have the lateral pressure gradient that appears when the flow turns–interact with the thin lateral edge of a plate. Theoretically, this allows to further reduce the hydraulic resistance, as well as the load on the plate.

The thin surfaces of the device completely eliminate the initial roughness of the flow, thus preventing the shedding of large eddies, and they add virtually no additional perturbance to the flow. Noticeable eddy structures are only observed at the trailing edge of the plates, but they are minor and fade almost immediately. The hydraulic resistance coefficient of this conditioner was found to be 1.3.

In the course of the research, we have carried out the calculations for all the flow conditioners that are represented in Figures 3 and 4. The velocity profiles behind the existing flow conditioners are shown in Table A1 of Annex A, behind the newly devised ones – in Table A2.

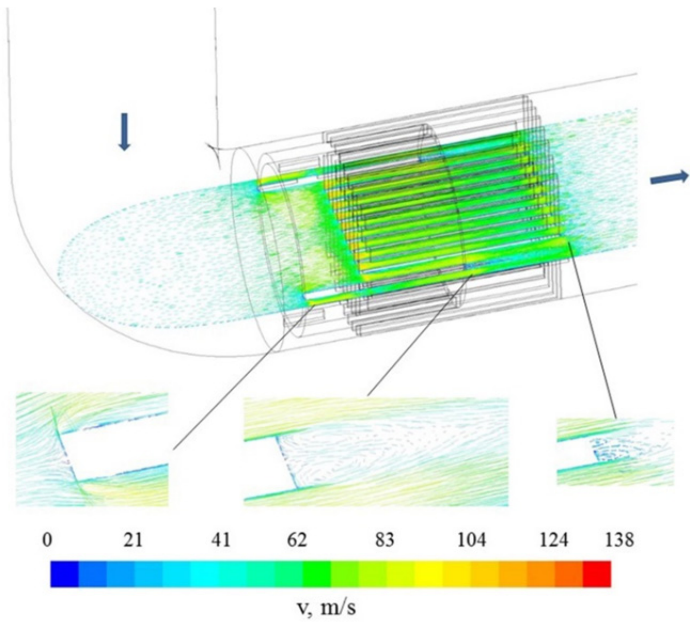


Figure 13. The flow in the channels of a plate-type flow conditioner (the diametral cross-section of the pipe, D_{in} = 100 mm, the plates are in the plane of the flow's turn)

4. The Assessment of the Efficiency of the Flow Conditioners

The installation of the flow conditioners of all of the examined types leads to a sufficient decrease of the roughness coefficient, which is very well illustrated by Figure 14. In comparison with the initial variant, the roughness of the velocity field, and, consequently, the dynamic loads decrease by at least 40%.

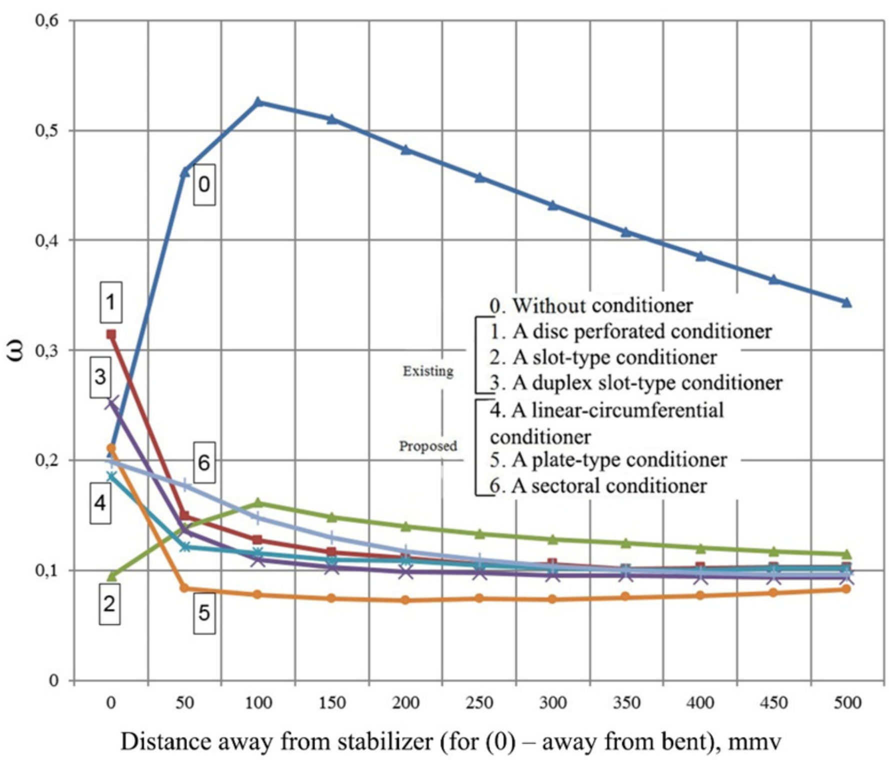


Figure 14. Comparison of the flow stabilizers in terms of the non-uniformity factor

The least effective proves to be the slot-type conditioner – due to its high coefficient of perforation. The curve of its roughness coefficient in Figure 14 is similar to that of the initial flow, which means it does not have enough effect on the eddy currents. However the duplex slot-type conditioner, while having the same coefficient of perforation, is free from this fault, because the bend of the flow between the two grates provides additional destabilizing effect on the eddies. Hence, the duplex slot-type conditioner is the most effective one of the currently used types.

It is worth mentioning that at the distance to the conditioner of approximately 3 diameters of the pipeline, the device's type ceases to have a significant effect on the roughness of the flow coefficient–for the disc, slot-type, duplex slot-type, linear circumferential and the plate-type and sectoral conditioners equally, its value becomes the same, 0.1, and further remains unchanged. This is due to the fact that the eddies, split by the conditioners, have low energy and fade fast. However, on the stretch before that point, the difference in the efficiency of the devices is significant and can have a crucial effect on the reliability of the examined system.

The results of the conducted research show that the plate-type conditioner is the most effective one, as regards smoothening the velocities profile, as the diagram in Figure 14 demonstrates. Its installation results in a decrease of the roughness coefficient by 56%, and at the distance of 250 mm to the stabilizer, this indicator is equal to 65%. However, it is comparable in the quality of smoothening with the most effective of the currently used devices – with the duplex slot-type conditioner, which is, additionally, a more compact one and more simple to produce. Still, the plate-type conditioner is the most effective one, because it has the least value of the hydraulic resistance parameter (Figure 15).

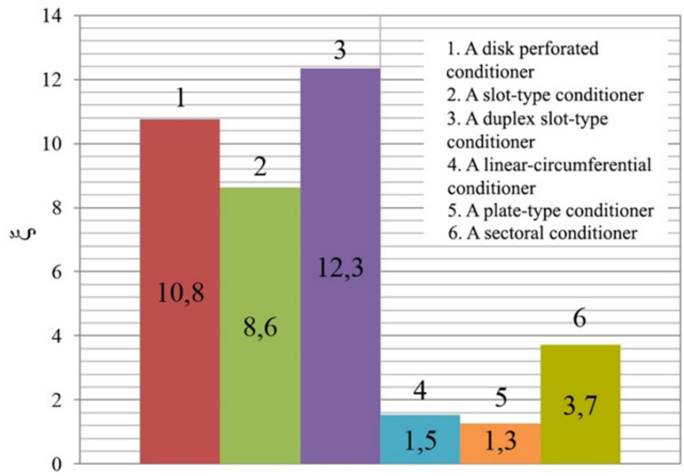


Figure 15. The comparison of the flow conditioners by the hydraulic resistance coefficient

Comparing the data, illustrated by Figures 14 and 15, we can conclude that the approach we employed to designing the conditioners of flow allows to achieve quality smoothing of the flow velocities profile, and, consequently, to reduce the dynamic loads, induced by it and affecting the equipment walls. The hydraulic resistance of the most efficient configuration turns out to be by 7÷8 times lower than that of the currently used types. The only and substantial disadvantage of the new flow conditioners, as compared against the existing ones, is being complicated in production. It is, maybe, advisable, then, to utilize the new devices in the high-pressurized pipelines, where the high value of the hydraulic resistance coefficient is crucial, from the point of view of energy losses. Moreover, in these conditions, the new flow conditioners will demonstrate better durability performance, since the area of the surfaces that experience the main difference in pressures is small.

Acknowledgments

The research has been carried out in Moscow Power Engineering Institute with financial support from the Russian Federation represented by Ministry of Education and Science of the Russian Federation under Agreement 14.574.21.0016 on Grant Provision dated June 17, 2014, for the purpose of implementation of the Federal Target Program "Research and Development in Priority Growth Fields of Russian Science and Technology Sector for 2014-2020". The unique ID of applied research: RFMEFI57414X0016.

References

Arshad, M. H., Kaharman, R., Sahin, A. Z., & Mansour, R. B. (2010). Second law analysis of compressible flow through a diffuser subjected to constant wall temperature. *International Journal of Exergy, 7*(1), 110-129. http://dx.doi.org/10.1504/IJEX.2010.029618

Bandyopadhyay, T. K., Ghosh, T., & Das, S. K. (2010). Water and air-water flow through U-bends – Experiments and CFD analysis. *Proceedings of the International Conference on Modeling, Optimization, and Computing, 1298*, 110-115. http://dx.doi.org/10.1063/1.3516285

Deitch, M. E. (1961). *Engineering Gas Dynamics*. Moscow, USSR: Gosenergoizdat.

Grigoryev, E. Y. (2014). *Development and research of vibration methods in annular diffusors of gas turbines.* (Unpublished candidate dissertation). Moscow Power Engineering Institute, Moscow, Russia.

Huang, G., He, Y., & Yu, J. (2012). Numerical simulation and vibration analysis of inner flow field for large-sized throttle valve. *Applied Mechanics and Materials, 233*, 154-157. http://dx.doi.org/10.4028/www.scientific.net/AMM.233.154

Karavosov, R. K., & Prozorov, A. G. (2014). On a hydrodynamic source of self-excitation of narrow-band disturbance in a wind tunnel. *Journal of Engineering Physics and Thermophysics, 87*(6)*,* 1480-1486. http://dx.doi.org/10.1007/s10891-014-1153-7

Khazaei, H., Mirzaii, I., & Teymourtash, A. R. (2010). Numerical modeling of turbulent flow in vortex tube. *Proceedings of the International Mechanical Engineering Congress and Exposition*, *7*, 257-266. http://dx.doi.org/10.1115/IMECE2010-40159

Kim, D. H., Chang, Y. S., & Jhung, M. J. (2014) Numerical study on fluid flow by hydrodynamic loads in reactor internals. S*tructural Engineering and Mechanics, 51*(6), 1005-1016. http://dx.doi.org/10.12989/sem.2014.51.6.1005

Kostiuk, A. G., Kumenko, A. I., Nekrasov, A. L., Kalinin, S. V., & Medvedev, S. V. (2000). An experimental analysis of pressure pulsations in the steam admission elements of a turbine installation. *Thermal Engineering, 47*(6), 529-537.

Laribi, B., Belkacemi, D., & Abdellah, H. (2012). Numerical investigation of turbulence models for free jet flow. *Applied Mechanics and Materials, 231*, 2082-2085. http://dx.doi.org/10.4028/www.scientific.net/AMM.229-231.2082

Li, S., Cain, S., Wosnik, M., Miller, C., Kocahan, H., & Wyckoff, R. (2010). Numerical modeling of probable maximum flood flowing through a system of spillways. *Journal of hydraulic engineering, 137*(1), 66-74. http://dx.doi.org/10.1061/(ASCE)HY.1943-7900.0000279

Lyubimov, A. K., & Shabarova, L. V. (2011). *Methods of calculation mesh building using Ansys ICEM CFD* (Methodical manual). Nizhny Novgorod, Russia: Nizhny Novgorod State University.

Martins, N. M. C., Carrico, N. J. G., Ramos, H. M., & Covac, D. I. C. (2014). Velocity-distribution in pressurized pipe flow using CFD: accuracy and mesh analysis. *Computers and Fluids, 105*, 218-230. http://dx.doi.org/10.1016/j.compfluid.2014.09.031

Pulat, E., & Ersan, H. A. (2015). Numerical simulation of turbulent airflow in a ventilated room: Inlet turbulence parameters and solution multiplicity. *Energy and Buildings, 93*, 227-235. http://dx.doi.org/10.1016/j.enbuild.2015.01.067

Tsinoglou, D. N., Koltsakis, G. C., Missirlis, D. K., & Yakinthos, K. J. (2004). Transient modeling of flow distribution in automotive catalytic converters. *Applied Mathematical Modeling, 28*(9), 775-794. http://dx.doi.org/10.1016/j.apm.2003.12.006

Zaripov, D. I., & Mikheev, N. I. (2014). Amplitude-frequency characteristics of acoustic oscillations in T-shaped channels. *Thermophysics and Aeromechanics, 21*(5), 601-608. http://dx.doi.org/10.1134/S0869864314050084

Zaryankin, A., Rogalev, N., Rogalev, A., Kocherova, A., & Strielkowski, W. (2014). Line summary of approaches for improving vibrational reliability of thermomechanical equipment and its interconnecting pipelines at thermal power plant. *Contemporary Engineering Sciences, 33*(7), 1793-1806. http://dx.doi.org/10.12988/ces.2014.410190

Appendix A

The Velocity Profiles behind the Flow Conditioners

Table A1. The currently used types of flow conditioners

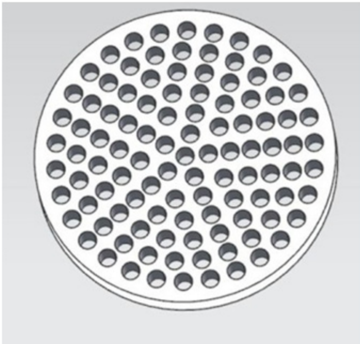
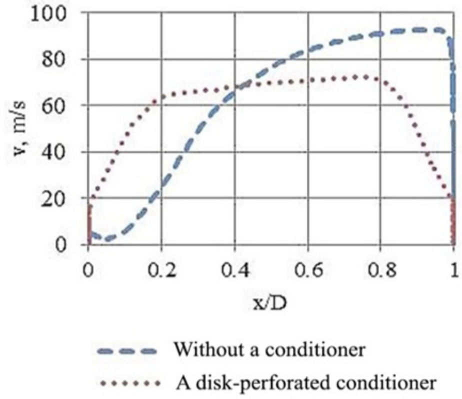
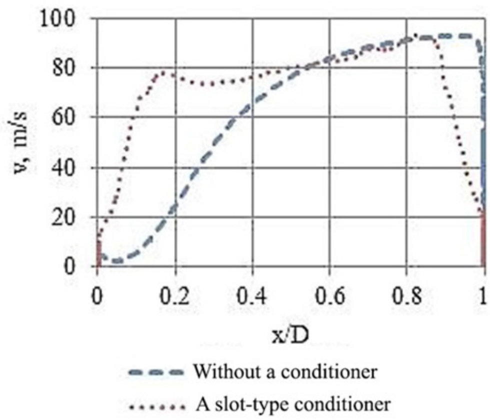
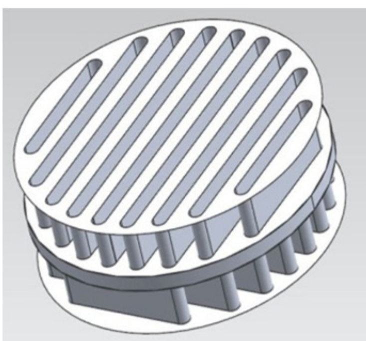
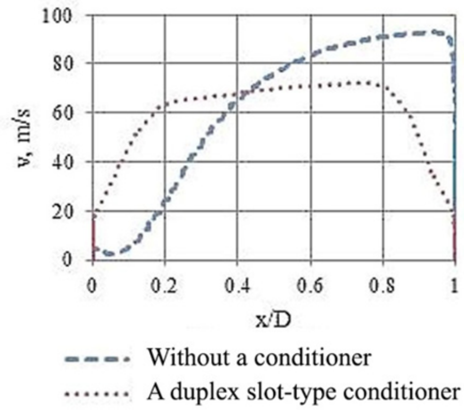
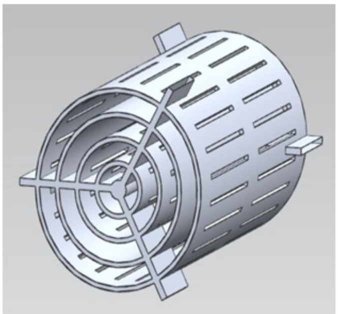
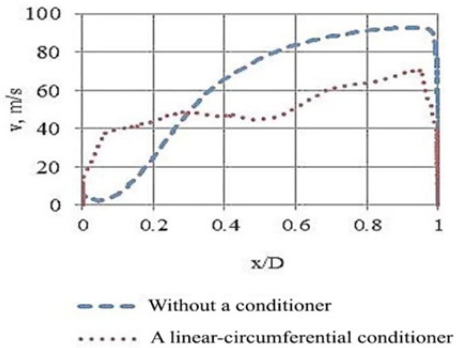
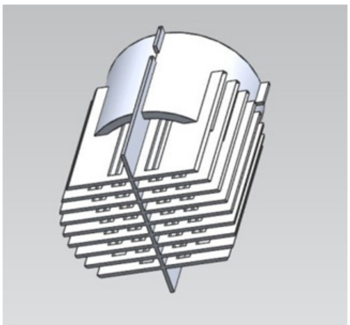
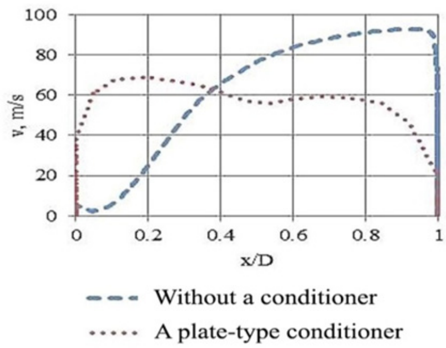
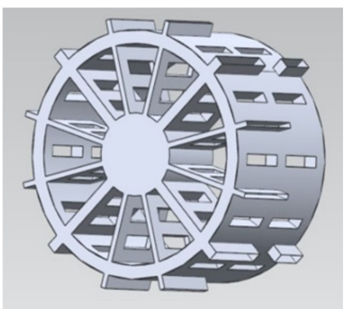
The type of the conditioner	The velocity profile
1. A perforated disk conditioner  $f = s = 0.35$	
2. A slot-type conditioner  $f = s = 0.42$	
3. A duplex slot-type conditioner  $f = s = 0.42$	

Table A2. The newly devised flow conditioners

The type of the conditioner	The velocity profile
1. A linear circumferential conditioner  *f* = 0.57; *s* = 0.78	
2. A plate-type conditioner  *f* = 0.35; *s* = 0.77	
3. A sectoral conditioner  *f* = 0.59; *s* = 0.62