Contemporary Engineering Sciences, Vol. 8, 2015, no. 31, 1485 - 1494
HIKARI Ltd, www.m-hikari.com
http://dx.doi.org/10.12988/ces.2015.59272

Application of New Flow Stabilizers to Reduce the Flow Non-Uniformity

Andrey Rogalev

National Research University “Moscow Power Engineering Institute”
Krasnokazarmennaya str. 14, Moscow, Russian Federation

Anna Kocherova

National Research University “Moscow Power Engineering Institute”
Krasnokazarmennaya str. 14, Moscow, Russian Federation

Vladimir Kindra

National Research University “Moscow Power Engineering Institute”
Krasnokazarmennaya str. 14, Moscow, Russian Federation

Ivan Komarov

National Research University “Moscow Power Engineering Institute”
Krasnokazarmennaya str. 14, Moscow, Russian Federation

Sergey Osipov

National Research University “Moscow Power Engineering Institute”
Krasnokazarmennaya str. 14, Moscow, Russian Federation

Abstract

This article researches the nature and reasons for occurrence of dynamic loads from the flow side on equipment walls, as well as considers existing flow stabilizers used to eliminate these loads. New designs of flow stabilizers are proposed based on analysis of disadvantages of these devices, and their efficiency is estimated in comparison with counterparts. According to the estimation results, the developed devices provide a comparable level of dynamic load reduction; however they have a considerably lower hydraulic resistance.

Keywords: power-generating equipment, steam distribution system of steam turbine, non-uniformity of velocity profile, dynamic loads, flow stabilizer

1 Introduction

Analysis of emergency situations at power plants and technologically connected sites show that the most widespread reason for failures of equipment, including thermal and mechanical units, valves and pipelines, is dynamic loads on equipment parts from the working fluid flow side. Those parts of equipment where a sharp change of the working fluid flow direction occurs are exposed to the highest loads. This is explained by effect of centrifugal forces which cause occurrence of a lateral pressure gradient by throwing fluid particles to the external (concave) walls of a bent channel [3, 19].

As a result, additional backgrounds for occurrence of separated flows are determined, that finally leads to formation of the non-uniform flow velocity profile. In turn, non-uniformity and transiency of a velocity field mean not the least rate of uneven pressure distribution in the flow. Therefore, in order to increase reliability of power-generating equipment it is necessary to thoroughly consider the nature of dynamic loads, as well as a structure of the flow leading to their occurrence, as well as to develop the efficient method of their elimination.

2 Nature of dynamic loads from flow side in bent channel

Time-varying pressure forces cause vibrations of the wide frequency range by effecting on the channel walls. According to results of an experimental research of pressure pulsation in steam-supply system of a turbine unit with initial nominal parameters of 24 MPa; 540°C under different conditions for low-frequency vibration amplitude testing (3.3; 7.5; 10.5 Hz) resulted on average about 2.5 MPa, in some cases reaching 4÷5 MPa. Pressure oscillation amplitudes in the spectral range of 400÷450 Hz reach 2.3÷2.5 MPa, and at higher frequencies (for instance, up to 1150 Hz) do not exceed 1.0÷1.2 MPa, however in this case the vibration energy may be significantly higher due to the high frequency [8, 11, 13, 14].

Such pressure oscillations have an apparently negative effect on reliability of equipment. In order to develop an efficient method of their elimination first of all it is necessary to consider a pattern of the flow causing such significant dynamic loads. Computational fluid dynamics (CFD) simulation is the most rational method of solution for this task.

The results of flow simulation in the bent channel representing a part of the steam distribution system of the turbine unit (Figure 1) show that large separated vortices are formed in places of the channel bending and cause disturbances at rather long pipeline sections [2, 7, 16]. When calculating, it was accepted that flows were isothermal, the pipeline walls (D = 100 mm) were adiabatic, output pressure was equal to atmospheric pressure, the flow rate was selected in such a way that the average flow velocity was 60 m/s, that is typical for power-generating equipment.

The flow velocity profile taken immediately following its turn show presence of separated flows and significant non-uniformity of the pressure field, as well. For comparison, the velocity profile is also given for the even flow existing in a straight pipeline at the same mass flow rate [12, 13].

It is known that the pressure pulsation frequency is directly related with the vortex sizes: the more the vortex size is, the lower the pulsation frequency and the higher the energy potential of pulsation movement. Large vortex flow areas cause the described above high-amplitude vibrations in the turbine steam supply system. As the vortex size is reduced their velocity in the flow approaches to the main flow velocity; the frequency of pulsations generated by them increases, and the total vortex energy sharply decreases. Therefore, the main way to reduce dynamic loads on the channel walls is to split large vortex cores into smaller formations and to evenly distribute them along the channel section [4, 5].

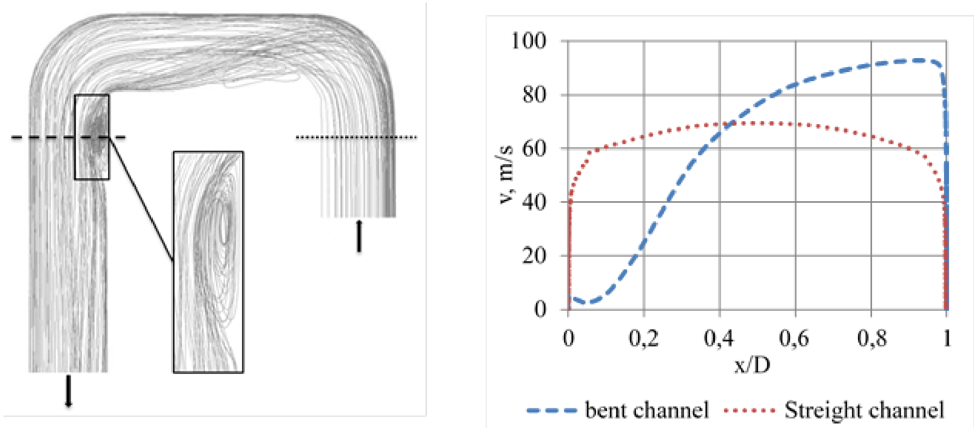


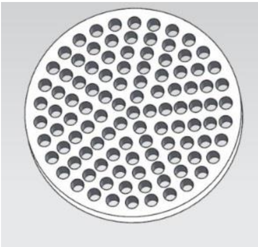
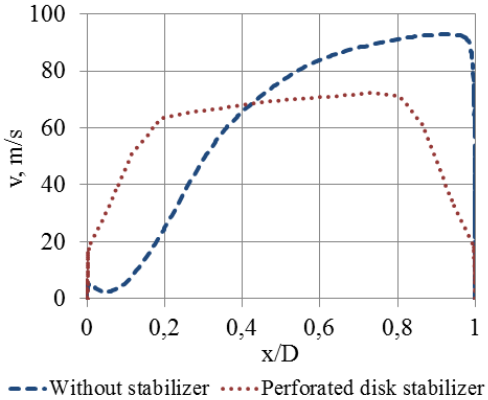
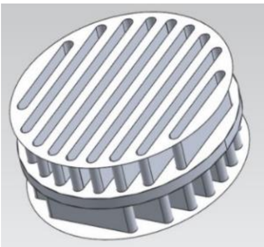
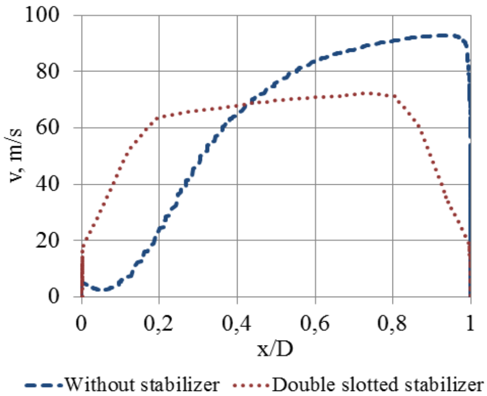
Fig.1: Flow in bent channel of steam supply system of steam turbine

3 Available flow stabilizers

The described above method of pressure pulsation elimination is implemented with so called flow stabilizers (aerodynamic filters, vortex breakers), which today are typically represented by perforated disks of different design installed in the pipeline. Table 1 gives the designs of the most common stabilizers and the results of their impact on the flow with the considered above curved element of the steam supply system of the steam turbine. On the average, they can reduce the pulsation amplitude by almost three times, and vibration displacement – by 2.2 times [19].

However, the installed flow stabilizer causes significant increase in hydraulic resistance of the system, since the flow area is lower in this case. Indices of hydraulic resistance of disk stabilizers are 9 and more, that despite all the benefits may make its use impractical [9, 10]. Thus, at the moment the actual problem is to develop flow stabilizers which not only efficiently reduce dynamic loads on equipment parts but also have low hydraulic resistance.

Table 1 – Common types of flow stabilizers

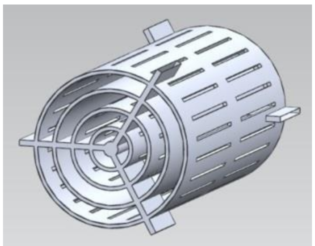
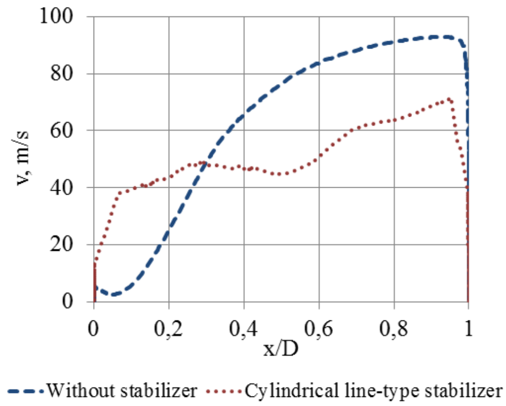
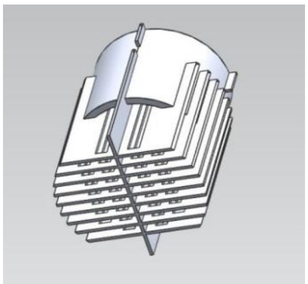
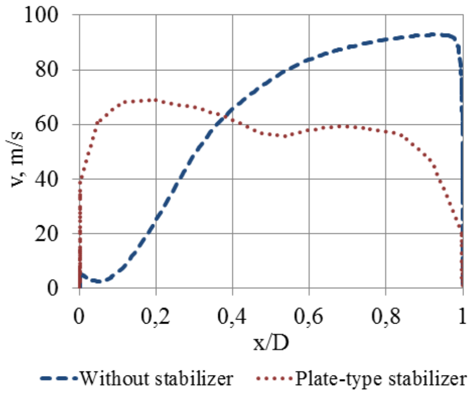
Stabilizer design	Velocity profile
1. Perforated disk stabilizer	
 *f* = *s* = 0.35	
2. Double slotted stabilizer	
 *f* = *s* = 0.42	

4 New flow stabilizers concept

In order to achieve the designated task it is necessary, first of all, to expand the flow area of the device. In this case the most efficient solution would be to reduce the stabilizer surface area disposed transverse to the flow direction, since vortices may be separated by means of the fluid interaction with thin surfaces situated in short intervals along the flow direction. Perforated stabilizer surfaces would also help to reduce hydraulic resistance, considering that, first, this would let the fluid flow freely under the effect of the lateral pressure gradient at flow deflection, and, second, this would reduce the differential pressure effecting parts of the device crosswise, therefore increasing its reliability.

Taking this into consideration, Moscow Power Engineering Institute supposed several design options for new flow stabilizers (Table 2).

Table 2 – Developed flow stabilizers

Stabilizer design	Velocity profile
1. Cylindrical line-type stabilizer	
 f = 0.57; s = 0.78	
2. Plate-type stabilizer	
 f = 0.35; s = 0.77	

The basic parameters of stabilizers include the following:

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

$$f = \frac{F_{hole}}{F_{\Sigma}}, \tag{1}$$

- the clear area index, equal to ratio between the flow areas before and after installation of the stabilizer:

$$s = \frac{S_{stab}}{S_{init}} \tag{2}$$

This index is introduced to describe the developed stabilizers with the main surface positioned along the flow, wherefore the perforation index cannot give any information on the flow area. In the considered case, calculation was carried out for a pipeline with the 100 mm inside diameter, however following the specified parameters, it' is possible to develop the similar stabilizer for any pipe size.

5 Flow stabilizers efficiency evaluation

Well-proven and widely applied CFD approach is used for research of different stabilizer design cases. A degree of velocity profile irregularity allow to estimate the intensity of dynamic loads on equipment walls in CFD simulation. Therefore, in order to evaluate the efficiency of flow stabilizers, it should be given a quantitative assessment.

An index of non-uniformity is very convenient to be used for this purpose, which is determined as follows [1, 17, 18]:

- index of non-uniformity for *i*-point of the velocity profile:

$$\omega_i = \frac{|u_i - \bar{u}|}{\bar{u}}, \tag{3}$$

where u_i – velocity value in i-point of the profile, $i = 1 \ldots n$;

$\bar{u}$ – average profile velocity range.

- index of non-uniformity for the velocity profile:

$$\omega_i = \frac{\Sigma\omega_i}{n}, \tag{4}$$

CFD simulation results show that installation of flow stabilizers of all considered designs allows to significantly reduce the non-uniformity index, which is well shown in Figure 2: in case of installation of the flow stabilizer the velocity profile non-uniformity, and therefore the dynamic loads, decreases at least by 40 %.

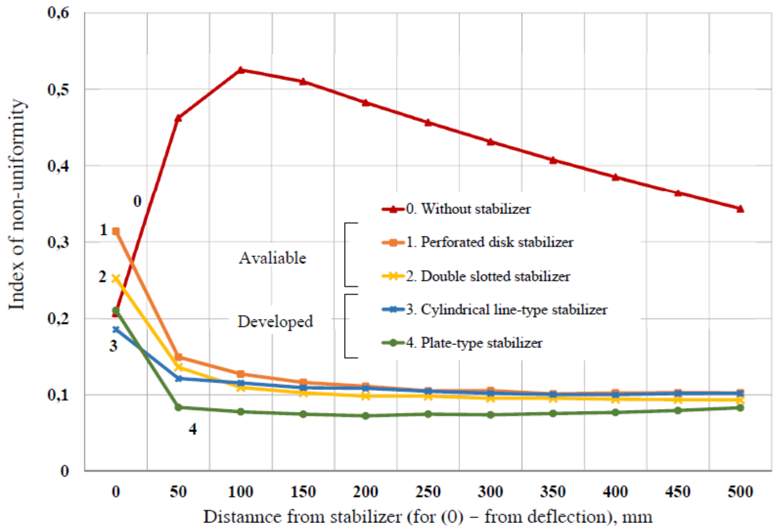


Fig. 2: Comparison of flow stabilizers by non-uniformity index

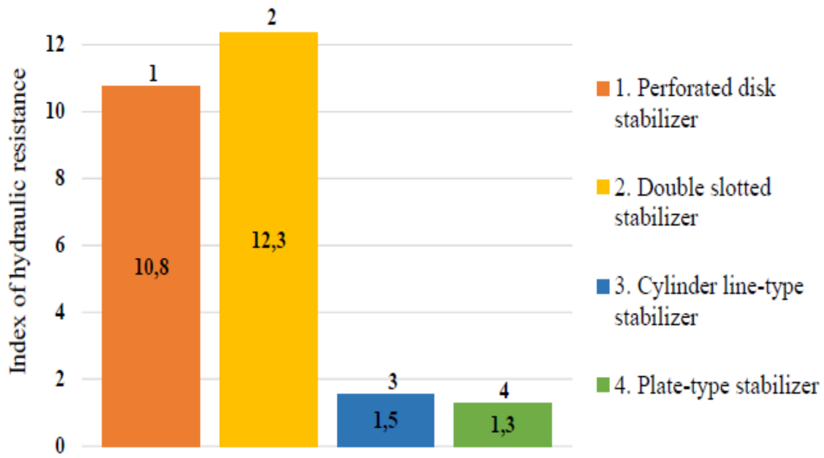


Fig.3: Comparison of flow stabilizers by hydraulic-resistance index

6 Conclusion

By comparing the data represented by Figure 1 and Figure 2, it is possible to conclude that the applied approach to designing of flow stabilizers allow to achieve quantitative leveling of flow rates and, therefore, reduction of dynamic loads on equipment walls effecting from the flow side. Moreover, the hydraulic resistance of the most successful design is by 3÷7 lower compared to the available counterparts. The only significant disadvantage of new flow stabilizers is their manufacturing complexity.

Acknowledgements. The research was made with financial support from Russian Federation represented by Ministry of Science and Education of The Russian Federation within the framework of the Agreement # 14.574.21.0016 on granting from June 17, 2014 for the purpose of implementation of the Federal Target Program “Researches and Development on Priority Directions of the Russian Science and Technology Complex for ears 2014-2020”. Unique identifier of applied research: RFMEFI57414X0016.

References

[1] M. H. Arshad, R. Kaharman, A. Z. Sahin, R. B. Mansour, Second law analysis of compressible flow through a diffuser subjected to constant wall tempera-

ture, *International Journal of Exergy*, **51** (2010), 110-129. http://dx.doi.org/10.1504/IJEX.2010.029618

[2] T. K. Bandyopadhyay, T. Ghosh, S. K. Das, Water and air-water flow through U-bends – Experiments and CFD analysis, *Proceedings of the International Conference on Modeling, Optimization, and Computing, Durgapur, India*, 2010. http://dx.doi.org/10.1063/1.3516285

[3] M. E. Deitch, *Engineering Gas Dynamics*, Gosenergoizdat, Moscow, 1961.

[4] A. M. Gotovtsev, *Development and Investigation of Steam Flow Stabilizing Systems in Exhaust Pipes and Remote Control Valves of Steam Turbines*, MPEI, Moscow, 2006.

[5] E. Yu. Grigoriev, *Development and Research of Vibration Reduction Methods in Annular Diffusors of Gas Turbines*, MPEI, Moscow, 2014.

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

[7] B. Ismailov, A. Urmatova, K. Ismailov, Mathematical modeling and calculation of dynamic characteristics of gas in multistage channels, *Applied Mathematical Sciences*, **7** (2013), 6571-6582. http://dx.doi.org/10.12988/ams.2013.310561

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

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

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

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

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

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

[14] N. Liu, G. Chen, X. Ge, J. Ning, G. Jiang, Cause analysis and solutions to the vibration of feedback system of steam turbine inlet valve, *Ethylene Industry*, **25** (2013), 41-43.

[15] A. K. Lyubimov, L. V. Shabarova, *Methods of Calculation Mesh Building Using Ansys ICEM CFD*, Nizhny Novgorod, Nizhny Novgorod State University, 2011.

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

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

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

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

Received: October 4, 2015; Published: November 19, 2015