



e-ISSN: 2319-8753 | p-ISSN: 2347-6710

IJRSET

International Journal of Innovative Research in
SCIENCE | ENGINEERING | TECHNOLOGY



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 13, Issue 12, December 2024

ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA

Impact Factor: 8.524

9940 572 462

6381 907 438

ijirset@gmail.com

www.ijirset.com

Structural & CFD Analysis of Air Release & Vacuum Break Valves

Damodhar Fattuji Rahate, Prof. Vaibhav Bankar

P.G. Student, Department of Mechanical Engineering, Vidarbha Institute of Technology, Nagpur, Maharashtra, India

Assistant Professor, Department of Mechanical Engineering, Vidarbha Institute of Technology, Nagpur,
Maharashtra, India

ABSTRACT: A mechanical air escape valve is mounted to avoid fluid from passing into the penstock without air. Hydraulic air release valves do not require external activation, but contain a mechanism moving through the penstock as fluid flows. Hydraulic exhaust valves are zero machine water leakage and can be used as protection mechanism in hydraulic systems. As the fluid moves into the tube, the device tends to reduce water head pressure on the turbine blades, and when the fluid starts moving the system blocks the valve opening such that the air passes into this pipe. This project will analyse the air release valve under dynamic fluid load conditions. This type of valve is mainly used as safety valve, turbine inlet valve, and pump valve for low to medium design pressures. They are operated by oil hydraulic systems for opening and closing or by closing weight and hydraulic pressure for opening. For turbine inlet valves, oil pressure can also be taken from the governor hydraulic oil system. The sealing system is of flexible, adjustable rubber/metal type to reduce leakage to a minimum. Water flow through the valve is possible in both directions.

KEYWORDS: Hydro Power Plants, Air Release Valve, Ansys workbench.

I. INTRODUCTION

Small hydropower plants (SHP) use energy production without problems and interruptions. Therefore, SHP architecture studies concentrate on secure and efficient service. A SHP's steady activity is the best condition as there is no improvement in the system's mechanical variables including discharge and pressure eye. Nevertheless, if the disrupted flow varies during hydropower production, a disruption may arise, triggering a rapid shift in system status. In the hydraulic conveyance device, including the penstock, flow parameters shift with time. It can cause severe high or low pressure in the penstock, and extremely high pressure may do considerable harm. Turbines, valves and other penstock equipment can be impaired. Even penstock can burst or collapse. There are major hydropower disasters triggered by water hammer, resulting in severe historical harm and loss of life. At Bartlett Dam and Oneida Hydroelectric Power Plant in the U.S., severe faults resulted in five deaths. Lapino SHP's penstock in Poland ruptured in 1997 after his new clearance tests. Finally, the penstock burst as a result of rapid valve closure at Oigawa Hydropower Station in Japan and three employees died. Load rejection, immediate load rejection and load acceptance conditions, turbine wicket or valve mechanical failure and abrupt shifts in front bay elevation will cause water hammer in small hydropower plants. To produce energy at a constant pace, a hydraulic turbine attached to a generator feeding a power grid must run at a constant rotational speed. Some frequency changes may affect generator and turbine velocities. Disruptions that alter turbine speed are regulated by the governor action that aims to hold the turbine at a synchronous rate by controlling the wicket gates in a Francis type turbine or by adjusting the position of the jet deflector and shutting or opening the needle valves in Pelton turbines. Both these immediate behaviors trigger turbine flow and other flow parameters, resulting in hammer pressure. The following fig shows: the tank airflow stops air lock & hydro- power plant configuration.

Aim and Scope

The main purpose of doing this thesis to study the fluid flow in the newly designed flow control valve. The aim of this thesis to maintain the pre-set flow of 5-6 LPM and showing the varies pre-set flows in the valve to use in the necessary conditions. Now, in any product development, the computational simulation is essential to validate with experimental results. This work is done to submit the paper for AIVA Engineering private limited to show CFD results on the flow control valve.

The main objectives of this thesis are:

- To build a geometry model of flow control valve.
- Computational Fluid Dynamics Simulation on FCV to study the flow velocity and pressure drop.
- Mathematical model of fluid flow in an orifice plate.
- Experimental testing on FCV to calculate the amount of fluid discharge.
- Validation of flow rate with CFD analysis,
- Theoretical calculations and Experimental results.

II. LITERATURE SURVEY

Anton Bergant, Amo Kruisbrink, Francisco Arregui studied on Dynamic Behaviour of Air Valves in a Large- Scale Pipeline Apparatus. This paper describes an experimental programme on the dynamic behaviour of air valves performed in a large-scale pipeline apparatus. Dynamic flow tests were performed at large (full) scale, since previous quasi-steady flow tests at small scale did not lead to realistic results. Investigations in a large-scale pipeline apparatus lead to a better understanding of the physical processes associated with the dynamic performance of air valves. Float type air valves of nominal diameter of 50 and 100 mm were tested in geometrically similar 200 and 500 mm test sections, to allow for the assessment of dynamic scale effects and the development of dimensionless parameter groups and dynamic scale laws. The approach in the determination of the dynamic performance of air valves Was to measure their response acceleration/decelerations, which are imposed upon the valve. In this way, the air valve behaviour following events like system start-up, pump trip and pipe rupture is simulated. Key results of the dynamic flow tests, including air release tests (valve slam) and column separation tests (effect of air valve on surge suppression), are presented and discussed.

Ali Göksenli, Barlas Eryürek studied on Failure Analysis of Pipe System at a Hydroelectric Power Plant. In this study, failure analysis of pipe system at a micro hydroelectric power plant is investigated. Failure occurred at the pipe system in the powerhouse during shut down operation of the water flow by a valve. This locking had caused a sudden shock wave, also called “Water-hammer effect”, resulting in noise and inside pressure increase. After visual investigation of the effect of the shock wave on the system, a circumference crack was observed at the pipe flange weld region. To establish the reason for crack formation, calculations of pressure and stress values at pipe, flange and welding seams were carried out and concluded that safety factor was high (2.2), indicating that no faulty design existed. By further analysis, pipe system and hydroelectric power plant was examined. After observations it is determined that the plant did not include a ventilation nozzle (air trap), that prevents the system of sudden pressure increase inside the pipes which is caused by water-hammer effect. Analyses were carried out to identify the influence of water-hammer effect on inside pressure increase and it was concluded that, according Jowkowsky’s equation, shut down time is effective on inside pressure increase.

Thong See Lee, Hong Tong Low, Dinh Tam Nguyen and Wei Rong, Avan Neo on Experimental Study of Check Valves in Pumping Systems with Air Entrainment. An experiment setup was introduced to study dynamic behaviour of different types of check valves and the effects of air entrainment on the check valve performance under pressure transient condition. The experiment results show that the check valves with low inertia, assisted by springs or small traveling distance/angle gave better performance under pressure transient condition than check valves without these features. Air entrainment was found to affect both wave speed and reverse velocity. With the increase of the initial air void fraction in pipeline, the experiment results show that the wave speed was reduced, the reverse velocity was increased. The first peak pressure increased initially and then decreased with the increase of the initial air void fraction, the pressure surge periods were increased proportionally with air void fraction due to the greatly reduced wave speed. The study can be applied to help choosing suitable check valves for a particular pumping system.

III. METHODOLOGY

FE analysis is used to find the behavior of the structure under the loading condition. The perfection in model is one of the factors influencing the accuracy of the solution.

i. Pre-processor

Pre-processor involves the choosing the materials that are involved in the analysis and entering the material data which are missing the engineering data. Meshing the model is another important process in the FEA, where certain condition has to be followed like aspect ratio and skew angle. Assignment of the material to the structure and parts, in case if the different part has different material. The predetermined boundary condition and the loading condition are imposed on the FE model.

ii. Solver

Solver is fully automated. It's split into pre-solver and post-solver. Pre-solver reads pre-processor code and formulates concept in mathematical representation. All the pre-processor process must be satisfied to create the mathematical model, and then the mathematical engine proceeds to form the elemental stiffness matrix for the given problem. Solver engine solves the problem and calculates to give the required result like stress, strain, displacement, temperature and frequency etc.

iii. Post-processor

In post-processor results obtained by the solver is read and interpreted. Animation and graph describe the effects in terms of deformation, pattern forms, and natural frequency. Stress obtained are analyzed by the stress plot which used know the maximum and minimum stress is present and by this we can conclude, weather the design is safe or not. Design modification can have carried out in those areas where the attention is required. It gives the representation of the stress, strain and deflection in any direction or in required direction also at any angle to the coordinate axis.



Fig : CAD model

The above shown valve model can have made with the help of Uni-Graphics software and it is in 3 dimensional cad model. Dimensions for the above model can be given below in the form of three dimensional geometrical model dimensions.

i. Meshing using by ANSYS software

Meshing can be done by use of ANSYS software. Here we used in meshing tetrahedral meshing can be used, Meshing is done by ANSYS software. In that meshing here we used tetrahedral elements. Linear tetrahedral elements are also steady stress elements with 4 nodes or linear stress elements with ten nodes. These elements are formulating in 3 - dimensional freedom with 3 degrees of freedom per node, these are the translational degree of freedom in the X, Y and Z directions, correspondingly. Here we done meshing it has 125187 nodes and 73623elements.

C. Solver Packages Used

i. Unix Graphics Software

UG software is modeling software. By the use of this Unix Graphics software we modeled air release valve.

ii. ANSYS Software The meshed geometry in imp" file format is imported to Ansys solver for execution. The appropriate boundary conditions are applied and problem is executed for the given loads. The results are represented for Vanishes stress consideration. By the use of Ansys software we done meshing and analysis can be done. In meshing tetrahedral mesh is used.

FCV Outer body:

It is the initial part of the body which has the inlet pipe connection to the valve. The ball check is rested in this body when there is no flow.



Figure FCV part 1.

Length of the FCV outer body = 28 mm Diameter of the inlet = 12 mm
Thread pitch = ½ BSPT (1.814 mm)

Ball check:

It is rested in FCV outer body and plays the key role in FCV by making pressure drop and controlling flow. The return flow is stopped by this valve.

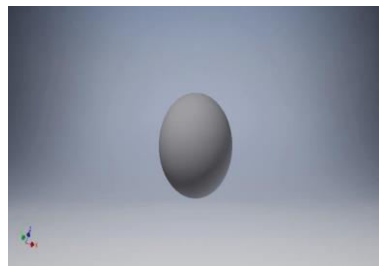


Figure FCV part 2.

Diameter of the ball = 12 mm Outlet Insert:

This outlet insert contains the outlet connection of the valve to the pipe and fixed to the FCV outer body. It helps to stop the ball while the flow runs.



Figure FCV part 3.

IV. EXPERIMENTAL RESULTS

The flow pattern in flow control valve can be shown in COMSOL Multiphysics through stream lines by adding a 3D plot. These stream lines show, where the fluid enters and comes out. As mentioned the flow type is turbulent, in Figure 6.1 the flow is having irregular fluctuations. The Figure 6.1 is kept in horizontal mode for the better vision of streamlines.

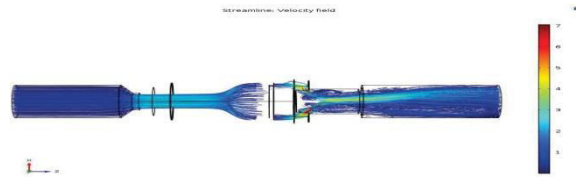


Figure Velocity Streamline profile.

The Figure explains the various angles and directions of fluid flow in the valve, the study this flow pattern is to make sure that flow is flowing around the sphere. In the Figure 6.2 its shows how the steam lines of the fluid pass through nozzles area and having higher velocity due to low area.

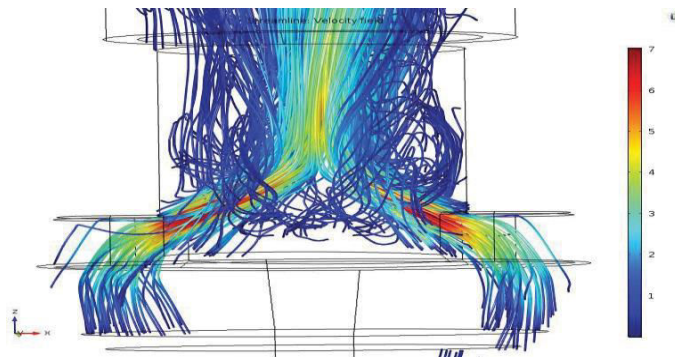


Figure Steam lines near nozzles.

The major pressure drop occurs in this region, where it is difficult to pass the fluid from the nozzles due to the small orifice dimensions. The highest velocity of the valve can be seen in Figure 6.2, the maximum velocity of fluid in the valve is 7.1 m/sec and it is located quite after the fluid passing through the orifice plate.

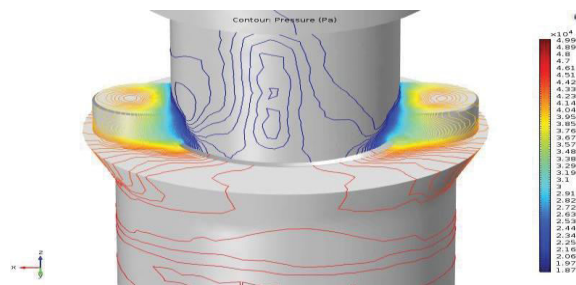


Figure Pressure drop near nozzles.

In the Figure, pressure drop near nozzles has shown clearly, the major part of the pressure drop occurs in this region. The interior part of the figure could show the perfect region and place of the pressure drop. Most of the pressure is dropped while the fluid is passing around the sphere. This helps to minimize the flow rate and keeps the pre-set flow to 5-6 LPM for the given boundary conditions.

V. CONCLUSION

1. Through the CFD simulations the perfect required design can be investigated and helps to manufacture the correct model which saves a lot of time and money.
2. Cavitation could possible to take place due to the empty space in the solid body and high pressure recovery than the vapor pressure of the valve.
3. From the investigation performed in this thesis it can conclude that the flow control valve can obtain a pre-set flow

of 5-6 LPM for the initial pressure of 0.5 bar.

4. The study of flow pattern shows that the fluid flows around the sphere and has the higher velocity near nozzles.
5. From now, it can confirm that use of Flow Control Valve is very suitable for household water distribution instead of using Ferrule valve.

REFERENCES

- [1] Lexus Hycon Pvt Ltd, 2014, "3 Way Flow Control Valve With Fixed Pressure Compensator".
- [2] P. Hydraulics, 2012, "Engineering Essentials: Flow-Control Valves,".
- [3] C. Helm, 2014, Autodesk 2014 Design and Creation Suites Give Flexibility to Innovate More. San Francisco: Autodesk, Inc.
- [4] AnsysInc,2016,"Ansys, <http://www.fluidcodes.com/archives/products/1445004823-ANSYS%20DesignModeler%20-%20Brochure.pdf>"
- [5] M. Eckert, 2006, The Dawn of Fluid Dynamics: A Discipline Between Science and Technology. Wiley.
- [6] P. N. Chandramouli, 2014, Continuum Mechanics. Yes Dee Publishing Pvt Ltd.
- [7] Comsol Inc, 2015, "COMSOL Multiphysics Modeling Software," <https://www.comsol.com/comsol-multiphysics#overview>.
- [8] FisherInternational,2005,"Valve'sReynoldsNumber, http://www.documentation.emersonprocess.com/groups/public/documents/catalog/cat12_s2.pdf"
- [9] SamsonAG,2011,"SizingandControlValves," https://www.samson.de/pdf_en/1351en.pdf.
- [10] G. Batchelor, 2000,Introduction to Fluid Mechanics.
- [11] OpenFoam,2004,"FluidFlow,<http://www.cfdsupport.com/OpenFOAM-Training-by-CFD-Support/node263.html>".
- [12] H. C. Buckland, I. Masters, J. A. Orme, and T. Baker, 2013,"Cavitation inception and simulation in blade element momentum theory for modelling tidal stream turbines," Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, vol. 227, no. 4, pp. 479–485.



INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

 9940 572 462  6381 907 438  ijirset@gmail.com



www.ijirset.com

Scan to save the contact details