SIMULATION OF HEAD LOSS IN TRASHRACK – A COMPARATIVE STUDY

[redacted]
jeethulakshmig@gmail.com

[redacted]
sumam@gectcr.ac.in

[redacted]
saji@gectcr.ac.in

ABSTRACT

Trashrack (TR) is a safe guarding structure which is provided at the inlet of the intake structure to prevent the entry of floating materials and boulders carried through flowing water into the water conveyance system. If these are not prevented from entering conveyance system, they may cause damages to the rotating parts of turbine unit of hydropower plants. Though it is an inevitable system to safe guard the rotating elements of power plant, it induces head loss when it is placed across the flowing water and hence, reduces the efficiency of energy production. Trashrack is normally provided as a grid of vertical and horizontal members. The effect of vertical member is significant than the other one and hence its effect is investigated here. The vertical members are available in different shapes among which rectangular profile bar is most commonly used. The present study focuses on the additional head loss contributed by the rectangular vertical bar profiles by considering Fluid Structure Interaction (FSI).

KEYWORDS

Trashrack, Rectangular profile, Head loss, 3D model, FSI

1.0 INTRODUCTION

Trashrack is an essential part of any intake structure. This is an underwater structure subjected to tremendous vibrations and causes considerable amount of head loss. Trashracks are located in the water intakes of hydropower plants to prevent the entry of large floating and submerged debris which could cause damage to the generating machine in general and to the pre-distributor, the distributor, the spiral-casing and runner of the turbine in particular. In addition it provides protection to boaters, swimmers, and the operating personnel and it may also prevent the entry of fish of larger size than the spacing of trashrack into the turbines. Trashrack can also be applied to the inlet of the intake structure of pumping station, water conveyance system, storm drain inflows and outflows and at the intake of water mill. Wood, steel, HDPE (High Density Polyethylene), FRP (fibre-reinforced polymer) etc. are the most commonly used materials for the construction of trashrack.

Generally adopted trashrack bar orientation is vertical. This is because horizontal orientation of trashrack bars provides poor performance and it also causes difficulty in the cleaning process. At the entrance of the penstock, the trashrack structure is attached to the top and bottom of the concrete structure. It consists of arrays of equally spaced parallel vertical bars held together by horizontal supporting beams and forms a grid. The spacing of the bars is determined by the maximum size of the body that is allowed to enter the turbine without clogging the distributor or the turbine runner.

Several studies have been taken place for the analysis of vibration and head loss present in the trashrack region and are discussed in the subsequent sections.

Trashracks in pumped storage systems with high flow rates can develop fatigue failures due to excessive vibration exerted by the flow past the rods in the rack.

Crandall et al., [1] conducted an experimental study of trashrack vibration on a half-scale model of a prototype rack design for the TVA Raccoon Mountain pumped storage system. The natural frequencies and loss factors of the first dozen natural modes of the rack were determined in air before placing the rack in a water channel. A half-scale model of the modified design was built and tested to verify the absence of destructive vibrations.

Many failures of trash-racks are dynamic in nature, and hence it is important to understand the dynamic characteristics of trashrack structures in general and a single rack in particular. Sadrnejad [2] introduced an accurate added-mass approach which can be employed to estimate the intensity of vibration of submerged structure.

Tsikata et al., [3] provides a study of turbulent flow near the trashrack models. In their study, the bar thickness, bar depth, and center-to-center spacing were maintained as constant. The flow characteristics were measured by aligning the bars with the direction of approach flow at three different stream velocities. The measurements were taken for four different bar inclinations relative to the direction of approach flow keeping the stream velocity constant. For each test condition, a high-resolution particle image velocimetry (PIV) technique was used. This study states that the relationship between head loss and bar inclination is nonlinear.

Trashracks represent an obstruction to the flow has to pass. The bars in the trashrack reduces the cross sectional area and forces the flow to accelerate through the gaps in the trashrack. This acceleration causes larger velocity and sheer stress along the bars. Due to the viscous effects, a thin layer called the boundary layer forms next to the surface. This creates a reverse flow region which is highly turbulent and unsteady, with shedding vortices and eddies that dissipate energy [4].

Fei and Xue-ping [5] studied the flow characteristics of the side inlet/outlet including water head loss, flow velocity distribution and inflow vortexes by numerical simulations. The ‘Volume Of Fluid’ (VOF) method is used for the simulation of incompressible, viscous and transient flow with free surface. The finite volume method is employed to discretize the governing equations.

The IS code IS 11388: 2012 [6] provides the standards for the design of trashracks. It describes the different types of trashracks and its selection criteria. The main sections in this code includes ‘inclinations of racks’, ‘velocity through racks’, calculation of ‘losses at trashracks’, ‘structural design of trashracks’, ‘structural details’ and ‘construction and maintenance of Trashracks’. It also included the available trashrack bar profiles and the standard formulas required for the design of the trashrack.

Hribernik et al., [7] has done an investigation into the different trashrack designs and their impact on fluid flow losses. They have selected three different rack bar profiles (one simple rectangular profile and two alternative aero-dynamically-shaped profiles) for the investigation which cause different flow losses. The flow simulation was done by 3D CFD simulations using an ANSYS CFX 12 solver. Gross head loss was calculated for each trashrack profiles and trashrack having minimum head loss was identified. The net profit were calculated and identified that the profit from the alternative trashrack design can be expected only after a period of 10 years.

Huang et al., [8] describes the effect of water in the dynamic response of large trashracks. These trashracks are prone to fatigue damage. To attenuate this problem, the trashracks are designed in such a manner that the coincidence between the excitation frequencies of vortex shedding and the natural frequencies of the trashrack are avoided. The adopted methodology includes the calculation of modal parameters and numerical simulation using finite element models, including the surrounding mass of water. An experimental investigation is also carried out by them for the validation of simulation by measuring the response using submergible accelerometers.

Josiah et al., [9] have proposed a new equation to estimate the head loss through trashracks of circular bars based on their experimental findings. A series of experiments were conducted to calculate the head loss through trashracks by considering various parameters such as inclination angle with channel bed, approach velocity, unit discharge and blockage ratio and finally arrived a new formula. This proposed formula proved to be giving head losses through trashracks with a reasonable accuracy but it is valid only for trashracks composed of circular bars and for partially submerged trashracks in uniform, steady flow conditions. To update the formulation, they found that fully submerged condition of trashracks as well as different bar shapes can be considered in estimating the head losses through trashracks. The study also extended to incorporate the effects of bed friction and channel slope on the estimation of total head loss through trashracks.

The effect of FSI on head loss is not taken in the earlier studies when a flow passing through the trashrack. This study analyses the effect of FSI on head loss for a trashrack with rectangular bar profile. A comparative study of these results has also been carried out with those obtained from the available empirical formula.

2.0 TRASHRACK BAR PROFILES

IS code 'IS 11388:2012' describes the details required for the design of the trashrack. As per this IS code, seven trashrack bar profiles exists and the shape factor (K) corresponding to each bar profiles are also defined. These 'K' values are used for computing the energy loss from empirical formula. Fig. 1 shows the available trashrack profiles and corresponding values of the shape factor.

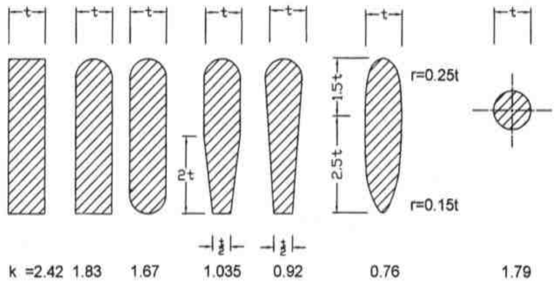

Figure 1 - Bar profiles as per IS 11388:2012.

Though seven trashrack bar profiles are available, this study considers only the rectangular bar profile.

2.1 Head loss Calculation Using Empirical Formula

The loss of head through the trashrack can be calculated from the following Kirschmer formula (IS 11388: 2012).

Head loss, $h_L = K\left[\frac{t}{b}\right]^{4/3} \times \frac{v^2}{2g} \times \sin\propto$ - (1)

2.2 Fluid Structure Interaction (FSI)

Fluid Structure Interaction (FSI) is characterized by the coupling between a structure and the surrounding fluid. The FSI process basically deals with the transfer of momentum and the forces to the structural part and the fluid surrounded to it. When the water passes through the trashrack it experiences a dynamic force. Hence the fluid and the structure cannot be treated separately like theoretical analysis; the interaction mechanisms are also to be considered. For simulating the fluid–structure interaction problems two main approaches are existing. One is 'monolithic approach' and other is 'partitioned approach'. The partitioned approach is again divided into 'one way coupling' and 'two way coupling'. In this analysis 'two way coupling' is considered.

3.0 NUMERICAL SIMULATION

The modelling is done based on an experimental setup where the water is flowing freely after passing through the trashrack. The fluid medium across the trashrack is water with density 998.2

kg/m3 and viscosity 0.001003 kg/m-s. Standard k-Ɛ (2 equations) model was selected for analysing the turbulence at the entrance of trashrack, since it requires less number of meshes compared with other models to represent the interspace between bars. The flow is “steady” and the solver type is selected as “pressure-based”. The “Gravity” (acceleration due to gravity) 9.81 m/s2 is provided in the z-direction in the general tab. The inlet boundary condition is taken as velocity 2m/s since the velocity range for the flow through the trashrack varies from 0.75m/s to 3m/s as per IS 11388:2012. The outlet condition is provided as atmospheric pressure since the water is falling freely. Work flow diagram for the numerical simulation is given in Fig. 2.

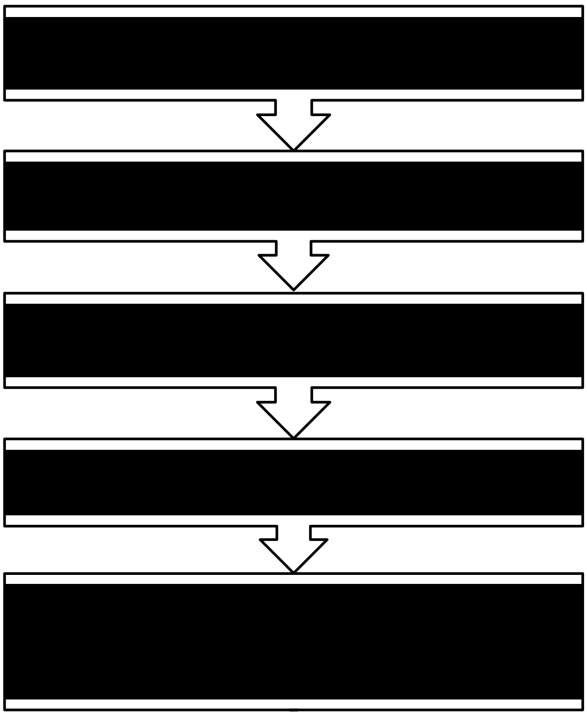

Figure 2 - Work flow diagram

The scheme selected is SIMPLE which is an algorithm that uses a relationship between velocity and pressure corrections to enforce the mass conservation and to obtain the pressure field. “Pressure-Velocity Coupling” is selected which uses a combination of continuity and momentum equations to derive an equation for pressure (or pressure correction) when using the “pressure-based” solver. Least square cell based method is selected for the spatial discretization of the model.

Important dimensions used for the modelling of a trashrack are:

- Thickness of vertical bar, t – 0.012m
- Width of vertical bar, d – 0.075m
- Clear spacing between vertical bar, b – 0.04m
- Clear spacing between horizontal bar, h – 0.4m
- Length of fluid body at upstream of the trashrack – 2*d
- Length of fluid body at downstream of the trashrack – 5*d

3.1 3D Modelling of a Rectangular Bar Profile

The dimension of rectangular bar profile is taken as 0.012m X 0.075m and the dimension of the surrounding fluid is the 0.052m X 0.6m. The dimension of surrounding fluid is chosen for the sake of simplicity of study of flow phenomena. For getting the 3D fluid domain, the property “Extrude” with the depth 0.412m is applied. The geometry is shown in the Fig. 3.

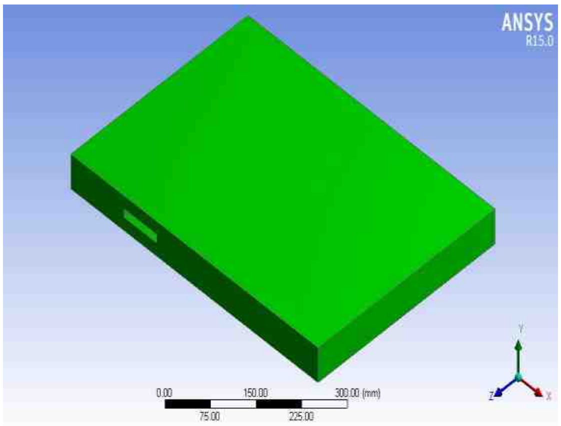

Figure 3 - Geometry of domain

The meshing of the geometry results in 1120530 elements. The head loss through the trashrack was analyzed by running the simulation on the geometry with the inlet velocity of 2m/s.

3.2 Analysis with Fluid Structure Interaction (FSI)

For incorporating Fluid Structure Interaction (FSI) with 3D model, the procedure for the preparation of geometry has been modified as follows. A small rectangle of dimension 0.012m X 0.075m was created. The property "extrude" in z-direction is applied to a depth of 0.412m and the structural part is created. A large rectangle of dimension 0.052m X 0.6m was drawn and extrude to 0.412m, and the required fluid domain was generated. The property "Boolean" with "subtract" operation is applied for keeping both the parts are independent.

The analysis is carried out for the newly created 3D model (Fig. 4) to incorporate the effect of Fluid Structure Interaction (FSI). For that in addition to the Fluid Flow (Fluent), Transient Structural and System coupling is used for the simulation.

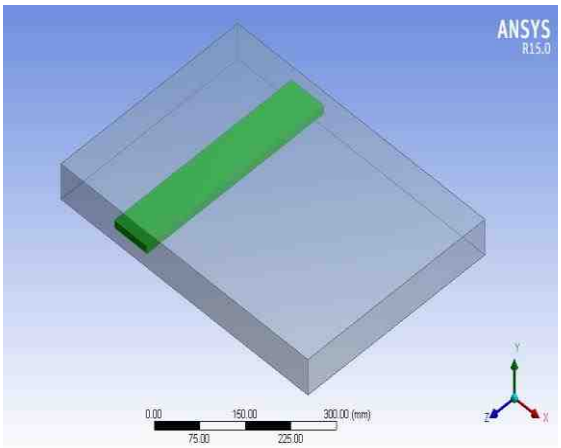

Figure 4 - Geometry for analysing the effect of FSI.

4.0 RESULTS AND DISCUSSIONS

The modelling of trashrack bar profile is done only for a single vertical member. For getting the effect of other vertical members, the property 'periodicity' is applied on two side faces of the model. For the better understanding of the analysis, two sections (Case I and Case II) were considered in geometry of the trashrack bar profile. The two sections are described based on the model having all the structural members.

Case I:

Section along the centre of the clear spacing between the two elements (By considering periodicity, the 'wallbottom' and 'walltop' will be act as a centre line through the clear spacing).

Case II:

Section along the periphery of the element

Representation of sections 'Case I' and 'Case II' are given the Fig. 5.

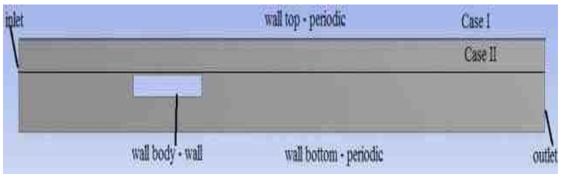

Figure 5(a) - The top view for the representation of sections Case I and Case II in the model used for the analysis.

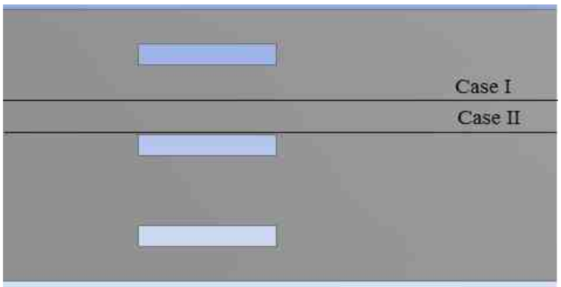

Figure 5(b) - The top view for the representation of sections Case I and Case II for a model having more than one vertical member.

If the "periodicity" is not provided on the either side of the fluid model with a single vertical member, both the sides will act as wall and effect of other members will not account for the analysis. On analysing the effect of periodicity based on the pressure value, the pressure value decreases on increasing the number of vertical members and reaches a minimum value for the model having highest number of members. On comparing the pressure value corresponding to the model with highest number of vertical members and the value from the model having single vertical member with periodic sides, it is found that the closeness of the results are very high. Fig. 6 describes the variation of pressure with increasing the number of members and the pressure on a single member applied with the property 'periodicity'.

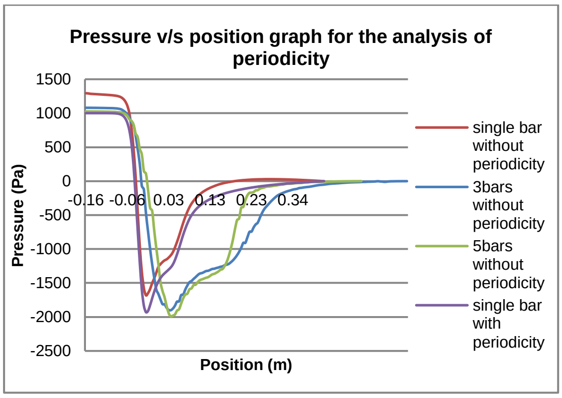

Figure 6 - Comparison of the effect of periodicity based on the pressure value.

4.1 Pressure and Velocity Analysis

From the inlet, the pressure value decreases towards vacuum pressure when the area of flow decreases in between structural member (due to periodicity) and increases towards the downstream end and finally to set pressure when the analysis done through the centre of clear spacing between the structural members (case I). From the normal value at the inlet, the pressure increases at the tip of the element and further decreases towards vapour pressure along the periphery to some distance (boundary layer) and then gradually increases to set pressure while the passing along the periphery of the structural element (case II). The pressure contour obtained from the simulation of 3D model is shown Fig. 7.

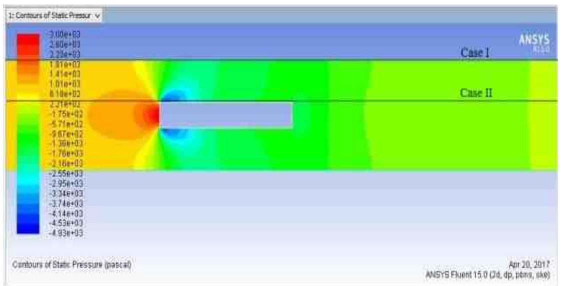

Figure 7 - Pressure contour in 2D plane at section Case I and Case II.

When the velocity passes through the centre of the clear spacing, from the inlet, the velocity increases to maximum when the area of flow decreases in between structural member (due to periodicity) and it starts decreasing towards the downstream end (case I). If the velocity variation through the section passing along the periphery of the structural element is analysed, it can be seen that from the inlet, the velocity decreases at the tip of the element and becomes zero along the periphery of the structural member to some distance (boundary layer) and then gradually increases towards the downstream end (Case II). Similar performance is observed in the study conducted by Crowe et al., [4]. The Figs 8(a) and (b) represents the velocity contour and the velocity vector respectively, obtained from the simulation of 3D model.

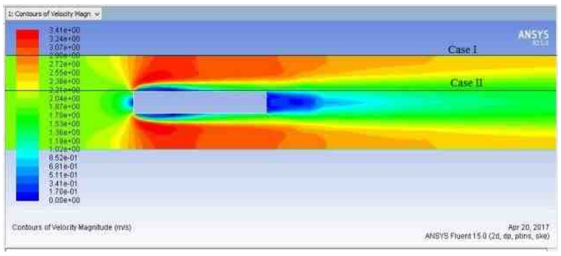

Figure 8(a) - Velocity contour in 2D plane at sections Case I and Case II

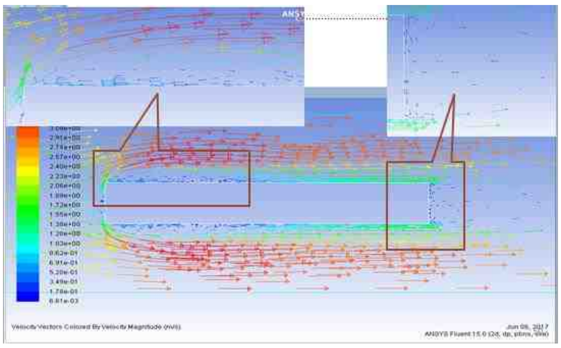

Figure 8(b) - Velocity vector diagram in 2D plane

4.2 Analysis of Head Loss

Head loss obtained from the simulation of the 3D model of a rectangular bar profile is $107.25*10^{-3}$m. The observed head loss shows an increasing trend for the fluid model and also for the model with FSI.

These results are compared with the results obtained from the empirical formula based on given shape factors (Table 1). It is found that the results obtained from empirical formula underestimate the actual head loss obtained numerically. This may be due to the interaction of fluid with the structure with sharp corners. These results are depicted in Fig. 9.

Table 1 - Comparison of head losses for 3D model, 3D model with FSI and Theoretical

Bar profile types	Head loss (m) for inlet velocity 2m/s		
	3D model	3D model with FSI	Theoretical
Rectangular (K = 2.42)	107.25* 10^{-3}	107.363* 10^{-3}	99.084* 10^{-3}

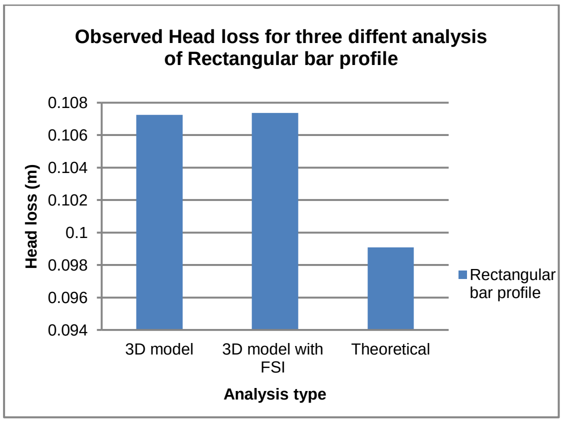

Figure 9 - Head loss v/s shape factor graph corresponding to 3D model, 3D model with FSI and Theoretical method

5.0 RESULTS

Comparative study has been done on the head loss obtained from the trashrack of rectangular bar profile based on the three analysis types (3D model, 3D model with FSI and Theoretical method). Numerical simulations were done on a 3D model applied with the property "periodicity", to reduce the size of the geometry and thus the software running time can be reduced, since the structure is symmetrical with x-axis. The pattern of pressure and velocity, along with the flow through the trashrack is analysed.

6.0 CONCLUSION

The head loss calculated from 3D model is lower than that from the model which considers the effect of FSI. FSI analysis captures the unaccounted head loss occurs in the simulation of 3D models. It is found that the calculated head loss using empirical formula is lower than that obtained from the numerical analysis (3D model and 3D model with FSI). i.e., the empirical equation with the given 'K' value underestimates the head loss for rectangular bar profile, though it is most commonly used in the trashracks.

NOMENCLATURE

h_L - Head loss through trashrack;
t - Thickness of bars;
b - Clear spacing between bars;
v - Velocity of flow through trash rack, computed gross area;
α - Angle of bar inclination to horizontal;
K - Factor depending on bar shape in accordance with Fig. 1; and
g - Acceleration due to gravity.

REFERENCES

[1] Crandall, S.H., Vigander, S. and March, P. A. 1975, "Destructive Vibration of Trashracks due to Fluid-Structure Interaction", *Design Engineering and Technical Conference*, 75, 1359-1365.

[2] Sadrnejad, S.A. 2002, "Hydrodynamics induced vibration to trash-racks", *IJE Transactions B: Applications*, 15(4), 357-366.

[3] Tsikata, J.M., Tachie, M.F. and Katopodis, C. 2009, "Particle Image Velocimetry Study of Flow near Trashrack Models", *Journal of Hydraulic Engineering*, 135(8), 671-684.

[4] Crowe, C., Elger, D., Williams, B. and Roberson, J. 2010, "Engineering fluid mechanics. 9th ed.", *Hoboken, NJ: Wiley.*

[5] Fei, Y.E. and Xue-ping, G.A.O. 2010, “Numerical simulations of the hydraulic characteristics of side inlet/outlets”, *Journal of Hydrodynamics*, 23(1), 48-54.

[6] IS 11388:2012, “Indian standard Recommendations for design of Trashracks for intakes (Second revision)”

[7] Hribernik, A., Fike, M. and Hribernik, T.M. 2013, “Economical optimization of a trashrack design for a hydropower plant”, *Journal of Trends in the Development of Machinery and Associated Technology,* 17(1), 161-164.

[8] Huang, X., Valero, C., Egusquiza, E., Presas, A. and Guardo, A. 2013, “Numerical and experimental analysis of the dynamic response of large submerged trash-racks”, *Computers & Fluids*, 71, 54-64.

[9] Josiah, N.R., Tissera, H.P.S. and Pathirana, K.P.P. 2016, “An Experimental Investigation of Head loss through TrashRacks in Conveyance Systems”, *Engineer*, 49(1), 1-8.

Presenting author Biodata

Name	:	Jeethulakshmi G
Designation	:	Student
Company	:	Govt. Engineering College,
		Thrissur - 680009
E-mail	:	jeethulakshmig@gmail.com
Mob	:	9495020257
Qualification	:	Pursuing M. Tech
Area of Expertise	:	Water Resources and Hydroinformatics

Significant Achievements: NA

Number of Papers Published in Journals: NA

Number of Papers Published in Conferences: NA