

Performance Evaluation of Impact Stilling Basin Using ANSYS Fluent



Ishan Sharma, Ashish Mishra, and Rakesh Mehrotra

Abstract Impact stilling basin or USBR type VI basin is a unique energy dissipator device that is installed at downstream of culverts, detention basin, and pipe outlets to prevent erosion and scouring. Its box-type structure with hanging baffle dissipates the energy of flowing water through impact thereby creating vertical eddies and consequently reducing the velocity of flow. In this study, a 3D model of impact basin is developed using ANSYS workbench, and its performance is analyzed in different scenarios using ANSYS Fluent R15.0 software. These different scenarios are taken as per those in published experimental work. Simulations are run for these cases in ANSYS Fluent by varying the position of baffle wall from both basin inlet and basin floor. Values of parameters such as turbulent kinetic energy, velocity of flow, wall shear stress and energy losses (kinetic energy and specific energy) are computed at the basin outlet for different cases to determine the position of baffle wall for maximum energy dissipation. Results from simulations show that maximum dissipation of energy occurs when the wall is placed at a distance of $4d$ from inlet with a gap of $1d$ from the floor, where ' d ' is taken as equivalent diameter of the pipe outlet. These results are consistent with the results of previous experimental studies.

Keywords Impact stilling basin · USBR type VI impact basin · Energy dissipator · ANSYS Fluent

I. Sharma (✉)

Department of Water Resources Development and Management, Indian Institute of Technology Roorkee, Roorkee 247667, India

e-mail: isharma@wr.iitr.ac.in

A. Mishra

Department of Hydrology, Indian Institute of Technology Roorkee, Roorkee 247667, India

R. Mehrotra

Department of Civil Engineering, Delhi Technological University, New Delhi 110042, India

1 Introduction

Energy dissipation is not only required at downstream of the spillway of a dam and barrage but also an important phenomenon to prevent erosion at canal outlets and culvert outlets [1]. This is crucial for effective water resources planning and management, which is emphasized in most countries of the present world [2–4]. The failure or damage of many culverts, pipe outlets, and detention basin outlet structures is caused by unchecked erosion. Energy dissipators are structures designed to protect downstream areas as well as drainage structures from erosion and scouring by causing energy losses and reducing the velocity of flow to acceptable limits. One such structure known as the USBR type VI impact basin was developed at the US Bureau of Reclamation laboratory in 1957 [5]. The basin is contained in a box-like structure, which is relatively small and does not require any tail-water to perform successfully. Energy dissipation takes place as flow strikes the vertical portion of the hanging baffle and then deflected upstream by the horizontal portion of the baffle and finally hitting the floor, thereby creating horizontal eddies [6].

An experimental study to reduce scour at downstream and to shorten the basin length of USBR type VI impact stilling basin was carried out by Goel and Verma [7]. For this, various experiments were performed in laboratory with Froude number ranges from 1.70 to 5.50. Wedge shape splitter blocks, end sill, and baffle wall of different sizes were used at different locations from pipe outlet. It was concluded that new models developed performed better than typical USBR type VI and also the length of the basin reduced to 25% with smaller tail-water depth.

Tiwari and Goel in their experimental study carried out 57 tests in laboratory for the enhancement of energy dissipation caused by impact stilling basin [8]. These experiments were performed to find out the effect of size, shape, and position of impact wall and gap between impact wall and basin floor. The performance was evaluated in terms of the performance index. The indices are simple techniques and widely used in water resources for assessing the performance or specific characteristics [9–11]. All the dimensions of the impact stilling were calculated in terms of equivalent diameter (d). It was concluded that size, shape, and location of impact wall affect the flow condition which ultimately affects the scour downstream of the basin. Higher the perform index higher the energy dissipation, and the highest performance index was observed with impact wall of size $1.5d \times 3d$ at a distance of $4d$ from the pipe outlet with length of basin reduced up to 30% as compared to USBR type VI stilling basin and with a gap of $1d$ between wall and basin floor.

Modeling and analysis of impact stilling basin has been done in ANSYS Fluent software [12]. This software uses numerical methods like finite element analysis (FEA) for simulation purposes. The process of numerical simulation involves three basic steps, viz. preprocessing, solver, and post-processing. The R15.0 version of

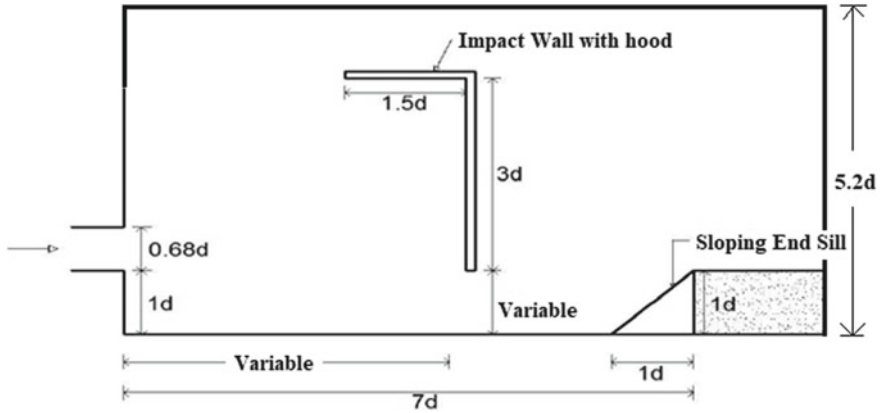


Fig. 1 Side view of the general basin model having dimensions in terms of equivalent diameter ' d '. *Note:* This figure is modified version of the original figure proposed by Tiwari and Goel [8]

ANSYS workbench is used which has its own software for drawing of geometry, i.e., design modular.

Reference Stilling Basin Model

Based on the experimental study carried out by previous studies [7, 8, 13, 14], six different models were developed and analyzed in ANSYS Fluent software in order to find the best model for energy dissipation downstream of a rectangular pipe outlet. These models were tested for three different scenarios, namely for Froude number = 1.85, 2.85, and 3.85, respectively, and the performance of stilling basin was evaluated by computing and comparing different parameters, viz. turbulent kinetic energy at outlet, kinetic energy and specific energy losses, and wall shear stress at outlet for different models in different scenarios. A general schematic diagram of stilling basin model is shown in Fig. 1 on which all the models were developed and analyzed in ANSYS.

2 Modeling in ANSYS

2.1 Calculation of Equivalent Diameter and Inlet Velocity

Equivalent diameter (d) is the most important parameter for designing an impact stilling basin as the dimensions of the basin (as well as model) is computed in terms of ' d ' (see Fig. 1). If the pipe outlet is circular in shape, the diameter of pipe outlet is taken as ' d ', but if the shape is rectangular or any other polygon, ' d ' is calculated by equating the area of that polygon to a circle. Say, for a rectangular inlet, ' d ' is calculated as the square root of $(4 * A / \pi)$, where A is the area of rectangle. In this

study, rectangular pipe of size 10.8 cm × 6.3 cm is used for analysis, therefore for each model the value of d is fixed as 9.3 cm.

The Froude number (Fr) at the pipe outlet is defined as $Fr = V/(gd)^{1/2}$, where V = velocity in m/s; g = acceleration due to gravity in m/s^2 , and d = diameter of the pipe. Therefore, for $Fr = 1.85, 2.85, \text{ and } 3.85$, the inlet velocity is computed as 1.767 m/s, 2.722 m/s, and 3.677 m/s, respectively.

2.2 Geometry

In this study, there are total six different models of impact stilling basin whose geometry has been drawn in design modular. All the models are drawn in shape of a rectangular box with length = $7d$ (65.1 cm), width = $6.3d$ (58.6 cm), and height = $5.2d$ (50 cm) where ' d ' is the equivalent diameter of pipe outlet ($d = 9.3$ cm). The geometry of six different models is shown in Fig. 2, and details of each model are given in Table 1.

2.3 Meshing

The meshing has been done in ICEM CFD software which is a part of ANSYS workbench. The meshing details of all the six models have been kept same. Tetrahedron method is used in assembly meshing with maximum eight inflation layers and growth rate of 1.4. Figure 3 shows the meshed model of the impact stilling basin.

2.4 Boundary Condition in Fluent Setup

The following steps are followed to set up the Fluent software for analysis: (1) In Fluent launcher 3D, *double precision* and *serial processing* option is selected and setup is started. In General tab, pressure type solver, absolute velocity formulation, transient time, and direction of gravity in $-ve Y$ -direction are selected. (2) Then in Models tab, multiphase, number of Eulerian phases 2, implicit scheme, and ' $k-\varepsilon$ ' viscous model are selected. (3) In the third tab, fluid from materials is selected, as air is already present new fluid water-liquid is added from database of ANSYS. Thereafter, primary phase (i.e., Phase I) to air and secondary phase (i.e., Phase II) to water are assigned. (4) Thereafter, boundary conditions and initial conditions are entered, inlet is taken as velocity inlet and velocity of flow entering the basin is entered and 'intensity and hydraulic diameter' specification method is chosen. Also Phase II volume fraction is taken as 1 to insure only water entered from the inlet. Outlet is taken as pressure outlet. (5) The solution is initialized using hybrid initialization method and after that liquid region is patched using patch option which ensures that

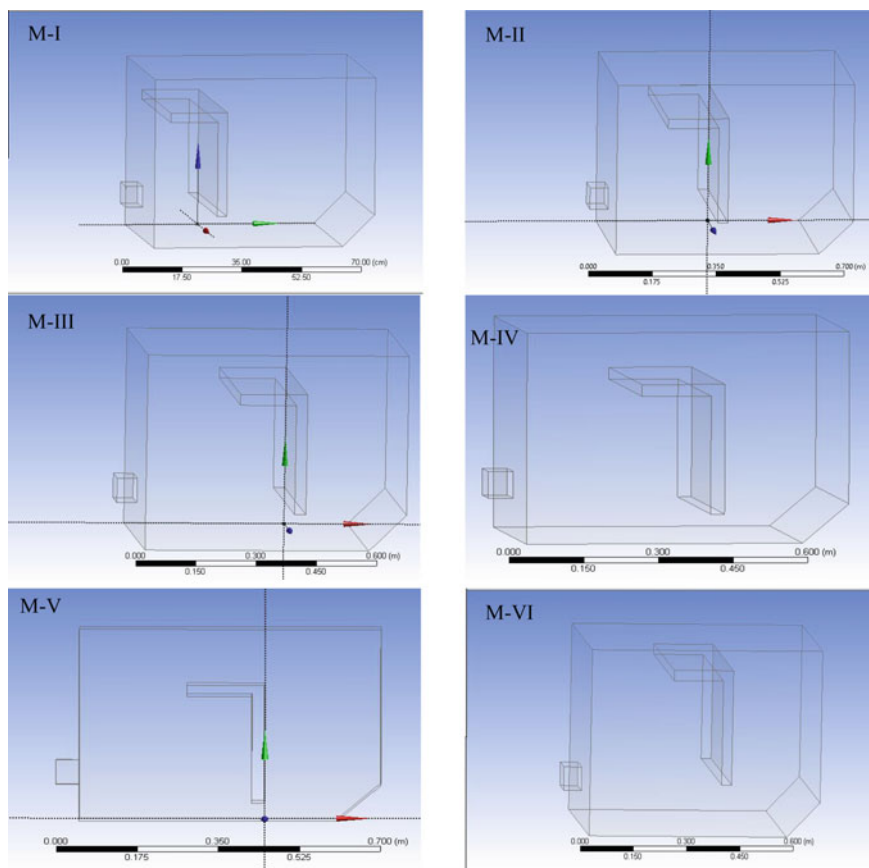


Fig. 2 Geometry of model M-I to M-VI drawn in design modular

Table 1 Various models designed in design modular by ANSYS Fluent software

S. no.	Model name	Impact wall size	Distance from inlet	Gap between wall and basin floor
1	M-I	$1.5d \times 3d$	$2d$	$1d$
2	M-II	$1.5d \times 3d$	$3d$	$1d$
3	M-III	$1.5d \times 3d$	$4d$	$1d$
4	M-IV	$1.5d \times 3d$	$4d$	$0.75d$
5	M-V	$1.5d \times 3d$	$4d$	$0.5d$
6	M-VI	$1.5d \times 3d$	$4d$	$1.5d$

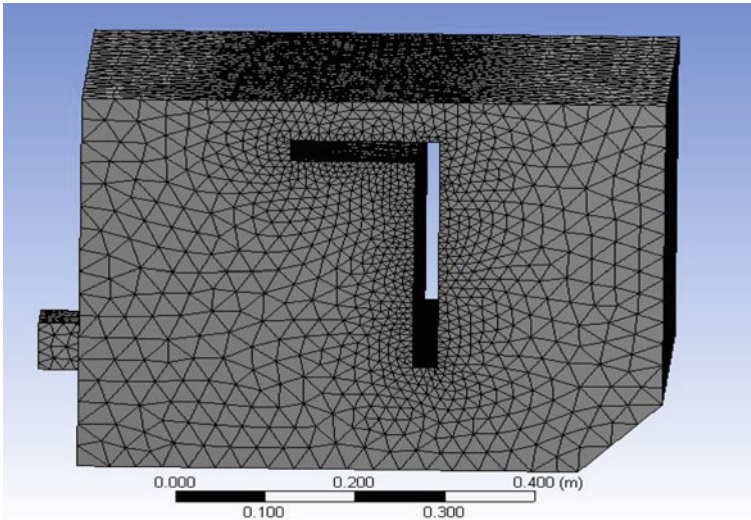


Fig. 3 Mesh diagram drawn in ICEM CFD

at the start of the simulation which portion is filled with water and which portion of basin is filled with air. (6) The last step is running simulations where time step size 1 s, number of time steps 40, and maximum iteration per time step 5 are entered and simulations are allowed to run.

3 Results and Discussion

After simulations are run for each model, the software builds stream lines plot for every model (Fig. 4) showing the simulated flow of water through the stilling basin especially after the water strikes the impact wall. As the distance between impact wall and inlet increases from $2d$ (18.6 cm) to $4d$ (37.2 cm), gap between impact wall and basin floor increases from $0.5d$ (4.65 cm) to $1.5d$ (13.95 cm).

The stream line plots from Fig. 4 give an idea of the velocity as well as the energy of the flow. The yellow and red color of stream lines denote high velocity, whereas blue and indigo colors signify low velocities; it also depicts the formation of eddies after the water strikes the impact wall as the distance between inlet and wall; the gap between basin floor and wall increases. Changes in the color of stream lines signify energy loss. Even though the stream line plots gave an idea about the energy losses and velocity vectors but the clear idea can only be known by computing the actual values of turbulent kinetic energy (TKE), kinetic energy (KE) and specific energy (SE) losses, and wall shear stress (WSS) at outlet. The results obtained from the analysis by Fluent are presented in Fig. 5 in the form of bar graphs comparing values

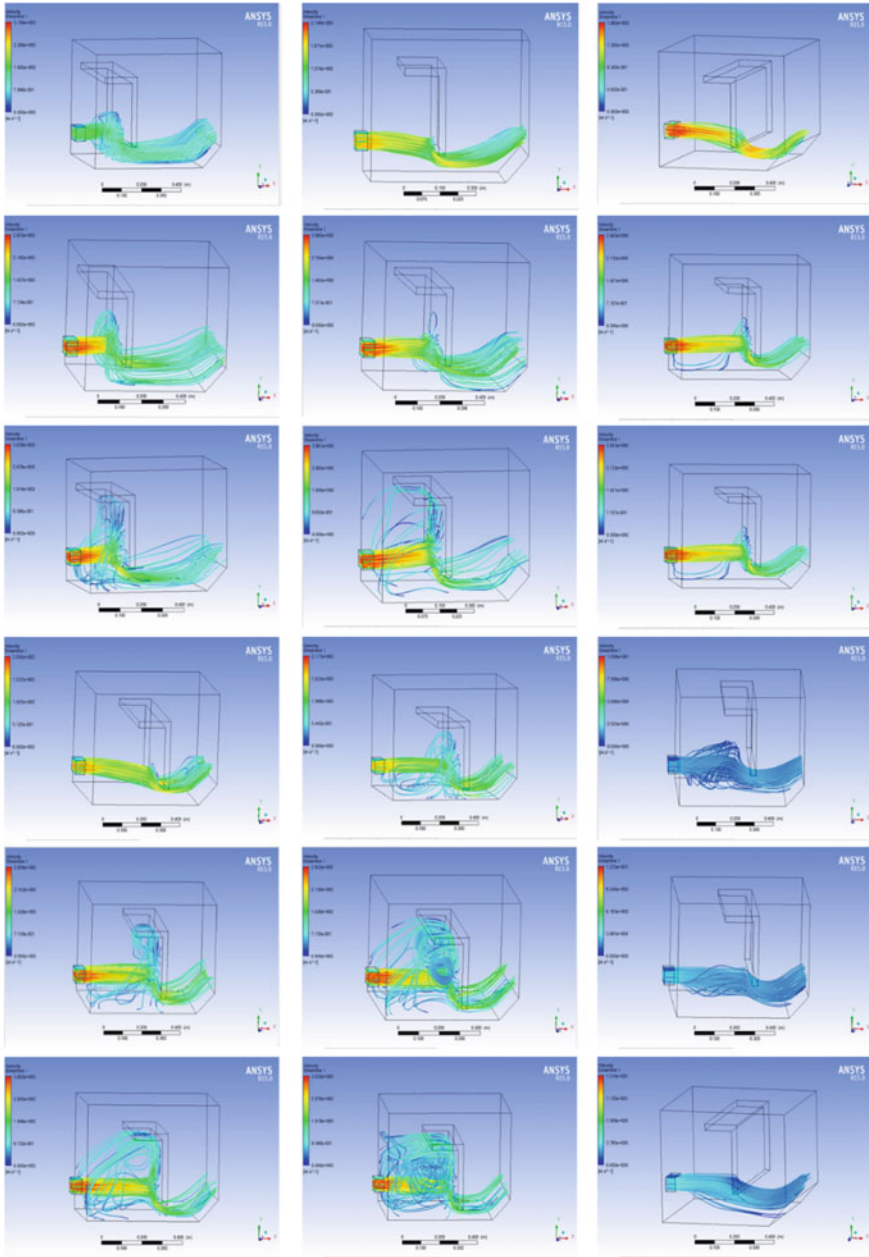


Fig. 4 Stream lines plot for each model (first row for the models M-I, M-II, and M-III corresponding to $Fr = 1.85$, second row corresponding to $Fr = 2.85$, and third row corresponding to $Fr = 3.85$; fourth row for the models M-IV, M-V, and M-VI with $Fr = 1.85$, fifth row corresponding to $Fr = 2.85$, and sixth row corresponding to $Fr = 3.85$)



Fig. 5 TKE, WSS, and the change in KE and SE between inlet and outlet for different models (M-I to M-VI) at different Fr (i.e., 1.85, 2.85, and 3.85)

of TKE, SE and KE losses, and WSS for different models with Froude numbers of 1.85, 2.85, and 3.85.

From Fig. 5, the following observations can be made:

For M-I model, the average as well as maximum value of TKE (in m²/s²) at the outlet of basin is highest (0.08 for Fr = 1.85, 0.104 for Fr = 2.85, and 0.126 for Fr = 3.85). This is because the impact wall is so close to the inlet that the jet of water strikes with more power causing more turbulence, which leads to the formation of horizontal as well as vertical eddies. This can be checked by seeing the stream lines



Fig. 5 (continued)

for M-I, M-II, and M-III model. For low Froude number (1.85), this effect is more prominent, but as the value of the Froude number increases ($Fr = 3.85$), this effect lessens.

For M-III model, the value of shear stress (in Pa) at bottom outlet is least (5.43 for $Fr = 1.85$, 5.52 for $Fr = 2.85$, and 6.26 for $Fr = 3.85$), and for M-I model, it is highest (5.61 for $Fr = 1.85$, 6.19 for $Fr = 2.85$, and 6.95 for $Fr = 3.85$). This may be because of the stream lines of the jet of water are partially hitting the wall causing more portion of jet to pass between the gaps, hence causing more shearing. This effect is more prominent for high values of Froude number.

Kinetic energy as well as specific energy loss (in m) is highest in case of M-III model (0.127 for $Fr = 1.85$, 0.331 for $Fr = 2.85$, and 0.634 for $Fr = 3.85$) and lowest in case of M-I model (0.109 for $Fr = 1.85$, 0.317 for $Fr = 2.85$, and 0.618 for $Fr = 3.85$). This may be attributed to the shearing effect and because eddies formation at the outlet is least in case of M-III model. The inferences are in agreement with the previous experimental study [8, 14].

For M-V model, the gap between the impact wall and basin floor is least (i.e., 0.5d); hence, the shearing area is least which causes value of TKE and WSS at outlet bottom to be more than other models (M-III, M-IV, and M-VI). But the value of above two parameters is least for M-III model and not M-VI model. This may be because as gap between impact wall and basin floor increases beyond 1d the jet of water that flows between this gap gets shear form one side only as the bottom portion of the wall not able to apply shear to the jet floe which can be seen from the stream lines plot.

4 Conclusion

From the different model simulations of impact stilling basin through ANSYS Fluent software, the model M-III (impact wall size $1.5d \times 3d$, the distance between inlet and impact wall = $4d$, and a gap of 1d between the basin floor and the wall) can be concluded as the best option. It produces less turbulent kinetic energy, less outlet velocity, less shear stress at the bottom outlet, and more energy losses, which lead to lesser chances of scouring in the downstream of the end sill.

References

1. Thompson PL, Corry ML, Watts FJ, Richards DL, Jones JS, Bradley JN (1983) Hydraulic design of energy dissipators for culverts and channels (No. HEC 14)
2. Swain S, Nandi S, Patel P (2018) Development of an ARIMA model for monthly rainfall forecasting over Khordha district, Odisha, India. In: Recent findings in intelligent computing techniques. Springer, Singapore, pp 325–331
3. Grafton RQ, Hussey K (eds) (2011) Water resources planning and management. Cambridge University Press, Cambridge

4. Swain S, Verma MK, Verma MK (2018) Streamflow estimation using SWAT model over Seonath river basin, Chhattisgarh, India. In: Hydrologic modeling, pp 659–665
5. Bradley JN, Peterka AJ (1957) Hydraulic design of stilling basins: hydraulic jumps on a horizontal apron (Basin I). *J Hydraul Div* 83(5):1–24
6. Vercruyse J, De Mulder T, Verelst K, Peeters P (2013) Stilling basin optimization for a combined inlet-outlet sluice in the framework of the Sigmaplan. In: IWLHS 2013-international workshop on hydraulic design of low-head structures, pp 55–66
7. Goel A, Verma DVS (2006) Development of efficient stilling basins for pipe outlets. *ASCE J Irrig Drainage Eng* 13(4):194–200
8. Tiwari HL, Goel A (2016) Effect of impact wall on energy dissipation in stilling basin. *KSCE J Civ Eng* 20(1):463–467
9. Swain S, Patel P, Nandi S (2017) Application of SPI, EDI and PNPI using MSWEP precipitation data over Marathwada, India. In: 2017 IEEE International geoscience and remote sensing symposium (IGARSS). IEEE, pp 5505–5507
10. Swain S, Mishra SK, Pandey A (2020) Assessment of meteorological droughts over Hoshangabad district, India. *IOP Conf Ser Earth Environ Sci* 491(1):012012
11. Swain S, Sharma I, Mishra SK, Pandey A, Amrit K, Nikam V (2020) A framework for managing irrigation water requirements under climatic uncertainties over Beed district, Maharashtra, India. In: World environmental and water resources congress 2020: water resources planning and management and irrigation and drainage, pp 1–8
12. Fluent, A. N. S. Y. S. (2011) Ansys fluent theory guide. ANSYS Inc., USA, 15317, pp 724–746
13. Tiwari HL (2013) Analysis of baffle wall gap in the design of stilling basin models. *Int J Civ Eng Technol* 4(4):66–71
14. Tiwari HL (2013) Design of stilling basin with impact wall and end sill. *Int Res J Recent Sci* 2(3):59–63