Numerical Analysis and Evaluation of the Energy Loss with an Altered Ski Jump Energy Dissipator: Maithon Dam, India

Shilpi Sippi Bhuinyan[1], Anand Kr. Sinha[2], Kiran Bhagate[3]
[1,2] Birla Institute of Technology, Mesra, Ranchi (Jharkhand), INDIA
[3]IDAC India Pvt Ltd., Pune (Maharashtra), INDIA
Corresponding Author Email: bhuinyan1984@gmail.com

Article Info
Page Number: 7598 - 7607
Publication Issue:
Vol 71 No. 4 (2022)

Article History
***Article Received**: 25 March 2022*
***Revised**: 30 April 2022*
***Accepted**: 15 June 2022*
***Publication**: 19 August 2022*

Abstract

One of the most significant features of a dam for flood protection is the spillway. Determining the hydraulic parameters of a spillway can be difficult due to the constantly changing flow type. For the objective of generating an effective design for the spillway of the Maithon dam in Dhanbad, Jharkhand, India, numerical model studies were conducted. An ogee spillway and energy dissipator models were examined in this study to see if they could improve energy dissipation and trajectory length, by using ANSYS Fluent 19.0.2D software models were developed for different profile with maximum discharge 13592 m^3/s. The numerical simulations using this design showed how the energy dissipator's design must be changed in order to provide proper energy dissipation and the best possible flow in the river downstream of the spillway. After going through numerical model simulations, a modified energy dissipator design in the shape of a ski-jump bucket with a wedge close to the lip was designed to reduce energy loss.

Keywords: Energy dissipation, Ansys- Fluent, CFD, Ski jump energy type Dissipator, numerical modeling, Ogee Spillway.

Introduction: Primarily, spillway flows are crest-based fast-changing flows with free surface streamline curvature. Two processes are taking place sequentially in the flow down the crest: the steady velocity and depth reduction of the main flow; the establishment and slow increase of the turbulent boundary layer; and the profile. Vertical acceleration defeats shear resistance in the flow at the solid boundary because of the quick changes in flow boundaries. The main difficulties in numerically solving the spillway problem are the quickly variable flow, the presence of both subcritical and supercritical flows, the creation of a turbulent boundary layer, an uncertain free surface, and air entrainment. Numerical simulation can be utilised to examine the behaviour of the complicated flow, even though spillway flow problems can involve both subcritical and supercritical flows. The Navier-Stokes equations used for model, almost any flow problem. Using simulation methods based on the Navier- Stokes equations and the proper approximations and assumptions, a range of flow difficulties have been addressed over the past 20 years. Valero D. and D.B. Bung (2015) and V. Yakhot et al. (1992) employed a full 3D technique based on the RNG k- ε turbulence model. The nonaerated region's point of origin and predicted velocity profiles were precisely located using this method. There is no one turbulence model that can adequately capture all turbulent difficulties, according to P. Bradshaw et al. 1996; Pope S.B. 2000; Wilcox D.C. 2006; and Hirsch C. 2007. Reynolds-averaged Navier-Stokes based 1D and 2D eqn. turbulence models are often used to calculate average forces, distributions, and velocity fields for hydraulic structures such as hydraulic jump stilling basins and stepped spillways, (P. Bradshaw et al. 1996). In their numerical simulations of a steep-stepped spillway as explained by Hirt and Nichols, F.A. Bombardelli et al. (2011) used RANS modelling with k-closure and a 2D-VOFsingle-fluid method (1981). A utilised a 2D k- ε turbulence model and a Lagrangian moving grid technique to track free-surface movement. Eghbalzadeh (2013) to replicate submerged hydraulic jumps. In order to examine 3D submerged hydraulic jumps computationally and compare their results to experimental data, Jesudhas et al. (2016) used the VOF approach in conjunction with detached eddy simulation. They conducted an examination that demonstrated the model's ability to predict the properties of a submerged hydraulic jump accurately. Numerous other researchers have examined hydraulic jump characteristics using numerical methods (e.g., C. M. Lemos et al. 2008; F. Ma et al.2001). Sills, chutes, and baffle blocks have been utilised by researchers to reduce the basin length for a design discharge and reduce tailwater depth (e.g., A.J. Peterka 1984; Fahmy S. F. Abdelhaleem 2013; Abdel Aal et al (2018). Numerical research was done on the turbulence properties of free and forced hydraulic jump by L. Qingchao et al. (1994). The FLOW-3D programme was used by Rostami et al. (2013) to simulate 2D undular hydraulic leaps. L. Qingchao et al. (1994), Ma et al. (2001), and Rostami et al. have all employed the concept of fractional volume of fluid (VOF) to admeasure the moving free surface (2013). D. Valero and D.B. Bung (2015) and V. Yakhot et al. (1992) were applied a fully 3D technique using the k- turbulence model and renormalization group.Even when simulating air entrainment with a verified subscale model, D. Valero, and R. Garca-Bartual (2016) study of aerated flows led to the conclusion that more work is necessary to precisely compute aerated zones; in fact, model on a smaller scale error may reach RANS modelling uncertainty. Carvalho R. F. et al. (2008) determined that using a 2D RNG k-ε model to simulate hydraulic jumps was sufficient despite variances in velocity and free surface profiles being detected.

Study Area: The current research examines the hydrologic features of the Maithon dam. Maithon dam is constructed across river Barakar near village Maithon in Dhanbad District Jharkhand (Figure 1). Maithon Dam is a composite earth and concrete structure. The catchment of Barakar river upto the dam site is estimated 6391.7 km^2.The maximum height and storage capacity of the barrage are 56.08m and 1093 MCM respectively. Spillway was designed for 13,592 m^3/s flood.The 12 bays that make up the spillway have radial gates that are 12.19 metres wide and 12.50 metres high.

Figure1: Location of Maithon Dam (source: Google Earth)

Data and information gathered from numerous pertinent sources are included for design and modelling. Original numerical model has been studied to get existing energy dissipation occurring at the site. Original numerical models have altered with different parameters to get maximum energy dissipation and validated with previous research work.

Numerical Analysis: The study demonstrated that it is possible to simulate transient basin flow structure and velocity profiles using CFD models. For free surface tracking, we use unstable Reynolds-averaged Navier-Stokes (RANS) equations with a renormalization group (RNG) k-ε turbulence model and VOF. Using CFD modelling (computational fluid dynamics), the equations governing mass, momentum, and energy conservation in a fluid flow have been solved. The turbulence stresses were modelled using the parameters of the submerged jumps. VOF, or Volume of Fluid, Method, created by C.W. Hirt and B.D. Nichols in 1981, is used to monitor the interface between two non-soluble fluids (air and water in our study).It employs an indicator scalar with a value range of 0 to 1 to indicate the fractional volume of the study's principal fluid, water.

Governing Equations: The incompressible flow is governed by the following continuity and momentum equations:

$\frac{\partial \rho}{\partial t} = \nabla.(\rho v) = S_m$ (1)

The mass conservation equation is expressed by VOF, or Volume of Fluid approach and the generalised form of Equation (1), which aids in the study of the VOF approach. For both compressible and incompressible flows, the Reynolds averaged Navier-Stokes (RANS) and k-ω shear stress transport (SST) equations are applicable. S (m) is the mass that has been moved from the dispersed second phase to the continuous phase (for example, as a result of liquid droplet absorption), v is the velocity of fluid, and is the fluid density.

An inertial (non-accelerating) reference frame conservation of momentum equation is Equation (2), which is defined as:

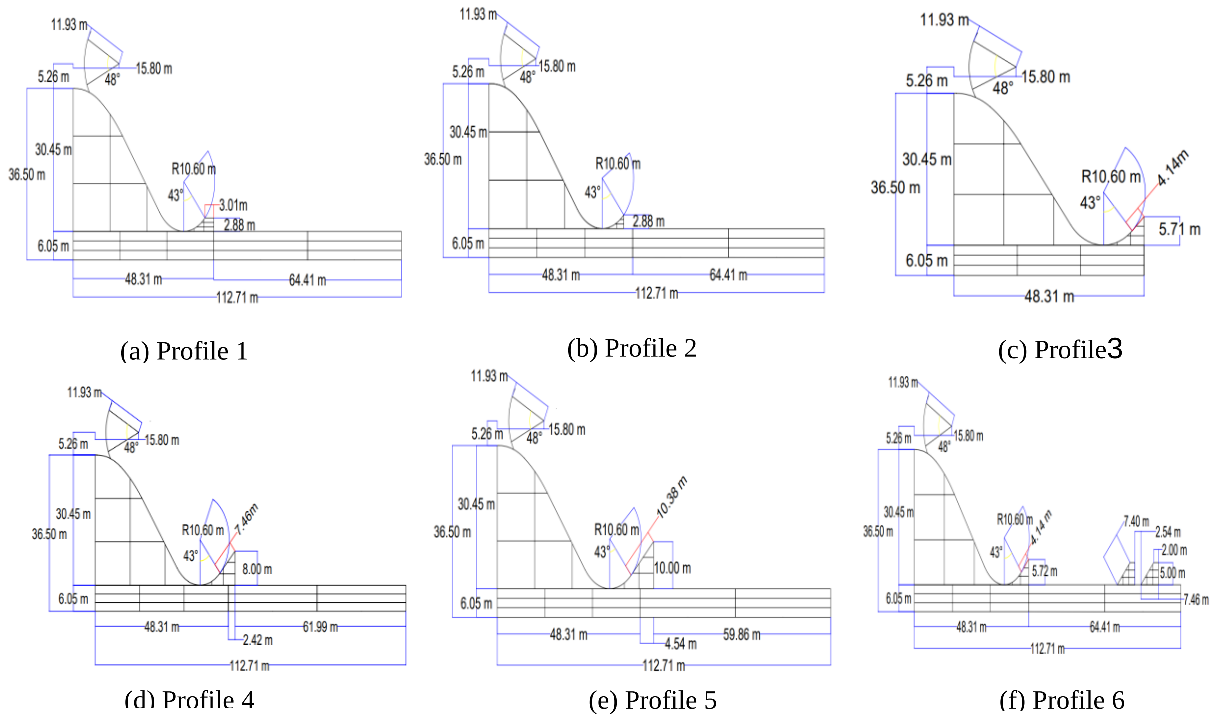
$\frac{\partial}{\partial t}(\rho\ \vec{v}) + \nabla.(\rho \vec{v}\vec{v}) = -\nabla p\ + \nabla.(\vec{\tau}) + \rho\vec{g} + \vec{F}$(2)

Where$\rho\vec{g}$ and $\vec{F}$ are the respective gravitational body force and external body forces, p is the pressure drop, is the stress tensor, and (e.g., those resulting from interaction with the dispersion phase). Equation is used to illustrate the stress tensor (3).

$\vec{\tau} = \mu\left[(\nabla\vec{v} + \nabla\vec{v}^T) - \frac{2}{3}\nabla.\vec{v}I\right]$(3)

Where, μ is the molecular viscosity, I is the unit tensor, and the second term on the R.H.S. is the effect of volume dilation.

In this study, the incoming discharge and/or velocity are unknown, but the reservoir water level can be determined at a flow entrance. This component should be situated far away from the spillway to prevent the reflective impact. The domain range should be used to determine the downstream border. For the analysis of the spillway crest and aerator region, the downstream condition will have no impact on the upstream flow because the flow over the spillway's downstream slope is supercritical. However, for the ski jump jet/hydraulic jump to be fully created, the downstream segment must be located far beyond the termination of the spillway. If the velocity and free surface profile are appropriately assumed, the computations will converge faster. The atmospheric pressure is term as operating pressure. The operating density is set at 1.223 m^3/s because air was the first phase to phase out of the two phases, air, and water.



(a) Profile 1 (b) Profile 2 (c) Profile3

(d) Profile 4 (e) Profile 5 (f) Profile 6

Figure2: Profile of Ogee Spillway and Ski type energy dissipator with different alteration.

Studies has been done for the bucket's initial design, which is depicted in Figure 4(a), for the maximum design of 13592 m^3/s with the gate fully open. Ski jump bucket type energy dissipator with radius 10.70 m were design and it was found that energy dissipation is about 50 %. Five distinct designs of the bucket were investigated in order to increase dissipation and minimize erosion.

Alternative designs for studies with a 13592 m^3/s maximum discharge were used. In alternative design shown in Figure 2 (b). there is no horizontal extension after bucket radius. In alteration profile 3, profile 4 and profile 5 shown in Figure 2(c, d, e), extension of length after bucket radius is 4.36m,7.46m and 10.38m to improve the energy dissipation at the downstream at the dam site. Profile 6 shown in Figure 2 (f), is the alteration of profile 3 with provision of wedges of size 5-

meter height, 2-meter horizontal width with 7.40 meter inclined with 45-degree angle to get maximum energy dissipation and control the erosion effect.

Result and Discussions: After performing the flow analysis in ANSYS Fluent, the numerical outcomes have been discovered. For a flow of 76 m^3/s/m, it can be observed that the stream performs a hydraulic jump in the stilling basin before entering the ski jump bucket, where it loses a lot of energy. Table 1 shows how the energy dissipation for various profiles may be easily seen.

Table 1: Profile Variation with Constant Discharge

Profile	q	V_1	Y_1	Y_2	X	E_1	V_2	E_2	% E.D.	Fr_1	Y_c
	m^2 /s	m/s	m	m	m	J	m/s	J	%	-	m
1	*76*	*24.15*	*2.22*	*1.3*	*25.7*	*31.94*	*17.11*	*16.22*	*49.22*	*5.17*	*8.38*
2	*76*	*24.21*	*1.79*	*1.84*	*70.1*	*31.66*	*16.25*	*15.30*	*51.67*	*5.77*	*8.38*
3	*76*	*23.8*	*2.52*	*2.48*	*48.5*	*31.39*	*12.07*	*9.91*	*68.44*	*4.77*	*8.38*
4	*76*	*23.51*	*2.54*	*2.91*	*50.3*	*30.71*	*14.70*	*13.92*	*54.66*	*4.71*	*8.38*
5	*76*	*22.18*	*2.59*	*3.01*	*72.7*	*27.66*	*17.14*	*17.98*	*34.99*	*4.40*	*8.38*
6	*76*	*23.8*	*2.52*	*4.13*	*48.5*	*31.39*	*9.40*	*8.63*	*72.50*	*4.77*	*8.38*

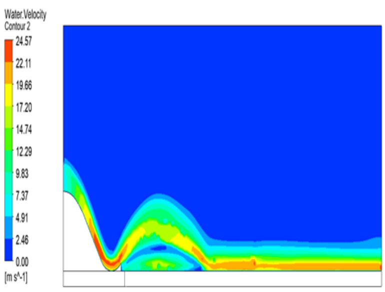
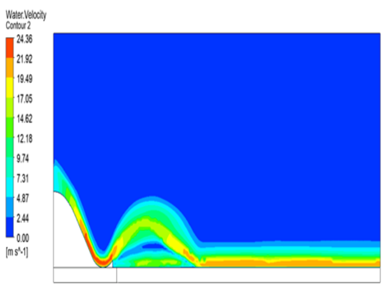
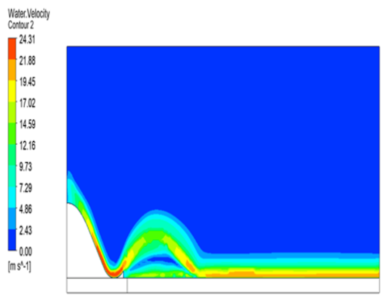
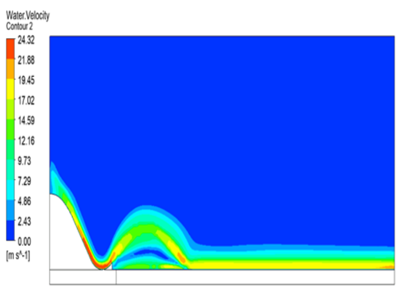
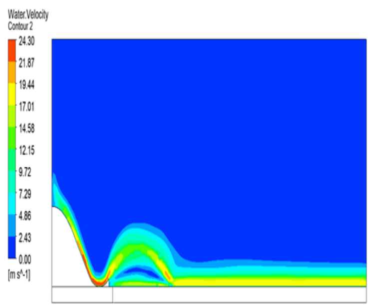
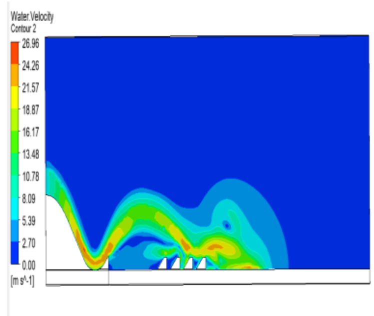
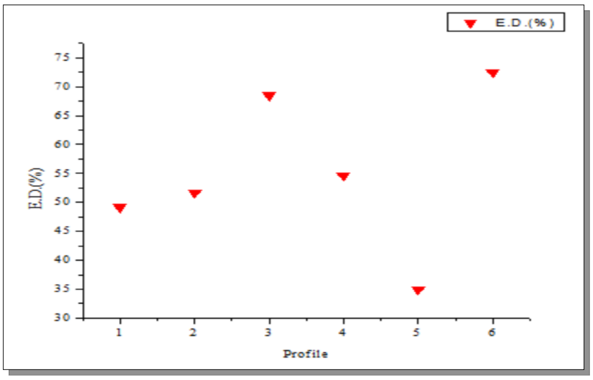

(a)

(b)

(c)

(d)

(e)

(f)

Figure 3: Water velocity contour of different Profile with maximum discharge

Figure 3 shows the water velocity contours on the ski jump bucket with maximum discharge. For various profiles, the water velocity on the bucket is significant. The maximum velocity occurs close to the bucket's length's midpoint, as can be shown for various profiles. Additionally, when the flow value increases, more velocity is exerted on the surface of the bucket. Also, for all quantities of the flow, it is possible to see the diminishing trend of the velocity near the lip of the bucket.



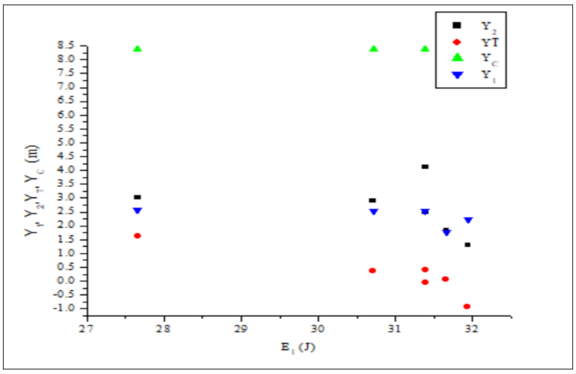
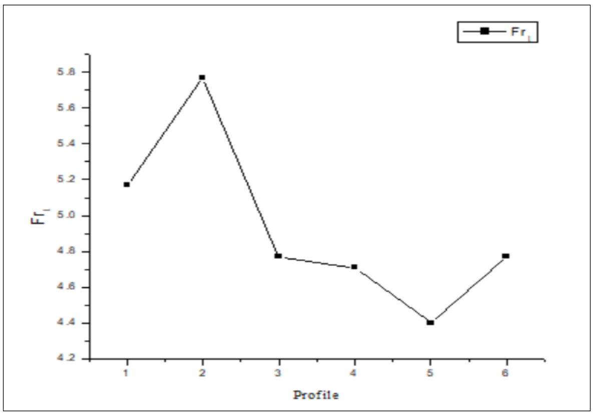


Figure 4: Energy dissipation Vs. Profiles **Figure 5: Depth. Vs. Specific Energy**

Figure 4represents that relation between Energy dissipation for different profile. Profile 1 has attained the loss of energy about 49.22 percentage while Altered profile gives more dissipation as compared to remaining profile, it indicates that profile 6 reduces the erosion at downstream side of the dam as compared to other said profiles.

Figure 5 shows that jump height curve and tailwater curve for varying specific energy. The jump water curve is above the tailwater curve for a maximum discharge indicating that the jump is away from the toe of the spillway.



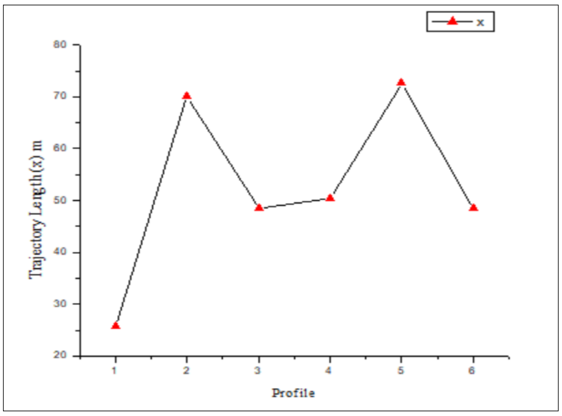


Figure 6: Froude No. Vs. ProfilesFigure 7: Trajectory length Vs. Profiles

Figure 6 represents the Froude no for different alternative models, it shows that Profile have Froude no range in between 4.5 to 5.77, hence there is a chance of steady jump formation. Figure7 shows that trajectory length of jet for different profiles. Profile 1 has a minimum trajectory length while profile 6 has a maximum trajectory length which indicates that there is less erosion occurring in profile 6 as compared to profile 3.

Acknowledgment: TheFirst author is grateful to the Executive Director (Civil), Damodar Valley Corporation, Maithon. The authors are really appreciativeto the Project Authority for permission to publish this paper and for providing data during the studies.

Abbreviations:

Q	Discharge, (m^3 / s)
q	Unit Discharge, ($m^2/s / m$)
E.D.	Energy Dissipation/ Energy Loss
R	Radius of Bucket
X	horizontal throw distance from bucket lip to the centre point of impact with tailwater, (m)
V_1	Initial Velocity, (m/s)
F_{r1}	Froud No.
Y_1	Initial depth before jump, (m)
Y_2	Depth after jump, (m)
Y_t	Tail Water depth, (m)
Y_c	Critical depth, (m)
E_1	Initial Specific Energy, (J)
E_2	Final Specific Energy, (J)

References:

1. Vimaldoss Jesudhas, Ram Balachandar & Tirupati Bolisetti, 2019, Numerical study of a symmetric submerged spatial hydraulic jump, Journal of Hydraulic Research, pp 1-14.
2. Gamal M. Abdel Aal, M. Sobeah, EsamHelal, M. El-Fooly,2018, Improving energy dissipation on stepped spillways using breakers, Ain Shams Engineering Journal 9,pp 1887–1896.

3. Valero D., and R. Garcia-Bartual, 2016, Calibration of an air entrainment model for CFD spillway applications, In Advances in hydro informatics, pp 571–582. Singapore: Springer.
4. Valero D., and D. B. Bung,2015, Hybrid investigations of air transport processes in moderately sloped stepped spillway flows, In Proc., 36th IAHR World Congress. Hague, Netherlands.
5. M. Javan, A. Eghbalzadeh,2013,2D numerical simulation of submerged hydraulic jumps", Applied Mathematical Modelling 37, pp 6661–6669.
6. Rostami, M. Shahrokhi, M.A. Md Saod, S.R. Sabbagh Yazdi, 2013, Numerical simulation of undular hydraulic jump on smooth bed using volume of fluid method, Appl. Math. Model. 37, pp1514–1522.
7. Fahmy Salah Fahmy Abdelhaleem,2013, Effect of semi-circular baffle blocks on local scour downstream clear-overfall weirs, https://doi.org/10.1016/j.asej.2013.03.003,Ain-shams-engineering-journal,Volume 4, Issue 4, pp 675-684.
8. BombardelliF. A., I. Meireles, and J. Matos,2011, Laboratory measurements and multi-block numerical simulations of the mean flow and turbulence in the non-aerated skimming flow region of steep stepped spillways, Environ. Fluid Mech. 11 (3): pp 263–288. https://doi.org/10.1007/s10652-010-9188-6.
9. CarvalhoR. F., C. M. Lemos, and C. M. Ramos.,2008, Numerical computation of the flow in hydraulic jump stilling basins, J. Hydraul. Res. 46 (6): pp 739–752. https://doi.org/10.1080/00221686.2008.9521919
10. Hirsch C.,2007, Numerical computation of internal and external flows: The fundamentals of computational fluid dynamics, Oxford, UK: Butterworth-Heinemann.
11. Wilcox D. C.,2006, Turbulence modelling for CFD, 3rd ed. La Canada, ˜ CA: DCW Industries.
12. F. Ma, Y. Hou, P. Prinos,2001, Numerical calculation of submerged hydraulic jumps, J. Hydraul. Res. 39, pp 493–503.
13. PopeS. B. ,2000, Turbulent flows, Cambridge, UK: Cambridge University Press
14. BradshawP., B. E. Launder, and J. L. Lumley,1996, Collaborative testing of turbulence models, J. Fluids Eng. 118 (2): pp 243–247. https://doi.org /10.1115/1.2817369.
15. L. Qingchao, V. Drewes,1994, Turbulence characteristics in free and forced hydraulic jumps, J. Hydraul. Res. 32, pp 877–898.
16.YakhotV., S. A. Orszag, S. Thangam, T. B. Gatski, and C. G. Speziale.,1992, Development of turbulence models for shear flows by a double expansion technique. Phys. Fluids A 4 (7): pp 1510–1520. https://doi.org /10.1063/1.858424

17. Peterka A. J.,1984, Hydraulic design of spillways and energy dissipators, Engineering Monograph 25. Denver: United States Dept. of the Interior. Bureau of Reclamation.
18. Hirt C.W. and Nichols, B.D.,1981, Volume of Fluid (VOF) method for the dynamics of free boundaries, Journal of Computational Physics, 39, pp:201- 225.