WATER QUALITY PARAMETERS PREDICTION OF HAORA RIVER SECTION BY MODELLING USING CFD

Dr. Bibhab Kumar Lodh
Department of Chemical Engineering, NIT Agartala
Email: bibhab.chemical.nita@gmail.com

ABSTRACT

The study focuses on the prediction of water quality parameters (WQP) in rivers, particularly during periods of drought, in order to minimize the impact of pollution. It utilizes computational fluid dynamics (CFD) as a valuable tool for developing detailed models that analyse the dispersion of WQPs in rivers.The research aims to establish the relationship between WQPs and key flow parameters. The turbulence model employed in the study demonstrates a strong correlation between parameters such as pH, dissolved oxygen (DO), biochemical oxygen demand (BOD), cadmium (Cd), iron (Fe), phosphate, and flow parameters including velocity, shear stress, and pressure. The software tool ANSYS Fluent is utilized for conducting the analysis, with AutoCAD used for geometry import, and MATLAB and artificial neural networks (ANN) employed for data validation.The specific study area investigated in this research is a section of the Haora River located in the state of Tripura. By comparing the results obtained from the CFD simulations with experimental data, the study demonstrates good agreement, with an error rate of approximately 15%.The overall goal of this work is to contribute to the understanding of how WQPs in rivers are affected by flow parameters, enabling better prediction and management of water quality during water scarcity events and aiding in the mitigation of pollution impacts. The study area is a section of Haora River in Tripura state.

Keywords: Water quality, River flow, CFD, ANSYS, ANN, Simulation

INTRODUCTION

General During many decades the growth of urban centres and industries occurred without any controls. The ramifications of this absence of proper coordination are experienced in all areas. Extensive research conducted at numerous research institutions has extensively examined and deliberated upon the impacts of human-induced actions leading to the contamination of water, soil, and air. People are taking notice of the risks involved in the misusage of natural resources. Thelikelihood of future freshwater shortages has risen, and certain areas are already experiencing the consequences. In several regions, people endure daily water rationing due to limited availability. These circumstances have sparked a heightened interest among industries and environmental agencies to invest in research initiatives and programs. The objective is to mitigate effluent emissions, anticipate the environmental consequences of new emissions, and improve wastewater treatment processes. Additionally, there is a focus on effectively managing and conserving water resources in order to address the growing challenges associated with water scarcity which already polluted bodies of water. However, major issues of river flow modelling are in the appropriate representations of the complex river flow conditions in the Computational Fluid Dynamics (CFD) model. Issues such as grid resolution, grid dependence, representation of wall roughness, appropriate turbulence models, etc., are all currently under intensive discussion. Nevertheless, with a numerical model and boundary conditions which provide adequate representations of the key processes of the river flow investigated, CFD simulations may provide considerable insight into, and clearer explanations of, the structure of the flow and the interactions of the key components of the processes than do the traditional field and/or laboratory measurements. The study of two and three-dimensional flow in open channels has experienced a surge of interest in the application of CFD to hydrological, and, geomorphological problems. The utilization of computational fluid dynamics (CFD) for complex flow problems has demonstrated significant potential, as indicated by studies conducted by Bradbrook et al. (2000) and Wu et al. (2000). Researchers are actively exploring the suitability of different turbulent models to effectively close the turbulence model. Another area of interest lies in the modeling of particle dispersion. Nokes and Hughes (2000) introduced a turbulent three-dimensional model to investigate the dispersion of particles in open channels with constant dimensions. Their approach involved applying a semi-analytic technique to analyze the continuous discharge of a non-degradable effluent in a channel with known velocity and diffusivity distributions. The model assumed the absence of secondary fluxes, highlighting a limitation in the original mathematical model.The primary contribution of this research lies in the proposition of a two-dimensional model capable of accurately predicting the dispersion of water quality parameters in open channels using CFD models. This advancement enables the prediction of substance dispersion over kilometer-long sections of rivers. Consequently, the outcomes of this study hold significant applications as a predictive tool, supporting and guiding management decisions, particularly in industrial settings.

LITERATURE REVIEW

General Technique by implementing a multigrid solver, and to investigate anisotropic turbulence modelling before moving on to modelling natural river Meselhe and Sotiropoulos (2000) presented early results obtained from their own investigation of open-channels using CFD. A simple numerical technique is used and an Alternate Direction Implicit (ADI) method is implemented for the solution of pressure. A two-point wall function is also used at the wall. The merit of their investigation is an attempt to account for the variation of the free-surface using previous time-step pressure information at the surface boundary. This means that it should be possible to investigate timedependent flow conditions such as floods using fully three dimensional models. Mesehle and Sotiropoulos tested their model against experimental data sets for a meandering in bank flow flume. Early results seem encouraging, although few details are provided. What is noticeable is that the use of the free-surface algorithm does not seem to enhance the quality of the solution compared to the rigid lid approach; however, the solution presented used an imperfect set of turbulence equations, the latter not being fully coupled with the free surface algorithm. The authors are well aware of such insufficiencies as underlined in their paper. In their conclusions they mention the need to improve the solution reaches. A contribution by Wu et al. (2000) has confirmed the rising interest from the civil engineering community for CFD techniques applied to rivers. This paper presents the results of a fully three-dimensional application in which a time dependent treatment of the free surface as well as a sediment transport model are included. The focus is on sediment transport. The model uses a simple k-□ model and the Stone method for the resolution of the discretized equations. The free surface is using outputs from a two-dimensional solution to update the mesh, an approach also used by the TELEMAC system. The sediment transport equations are based on the classic van Rijn formula. The later model is tested in fairly simple conditions and show good agreements with observations for the main morphological and sediment transport features. Detailed analysis reveals localised discrepancies that most probably steins from turbulence and from the sediment transport model, where the background science is much less accurate. Since no detailed hydrodynamics results are presented this is left to speculation however. The paper authored by Jae-Ho Jeong, Min-Seop Song, and Kwi-Lim Lee (2017) presents an intriguing possibility of investigating three-dimensional sediment transport features. The study introduces a practical methodology based on Reynolds Averaged Navier-Stokes simulation (RANS) for Computational Fluid Dynamics (CFD) analysis. The focus is on a real-scale 217-pin wire-wrapped fuel assembly of the KAERI PGSFR (Korea Atomic Energy Research Institute Prototype GenIV Sodium-cooled Fast Reactor). The main objectives are to support the licensing process for the KAERI PGSFR core safety and to understand the thermal-hydraulic characteristics within the fuel assembly. A notable aspect of this methodology is the innovative grid generation method, utilizing a Fortran-based in-house code with a General Grid Interface (GGI) function in the commercial CFD code, CFX. This grid generation technique enables the simulation of the fuel assembly's actual wire shape while minimizing cell skewness. The RANS-based CFD methodology is implemented with a high-resolution scheme for the convection term and the Shear Stress Transport (SST) turbulence model. The methodology is validated and expanded to other wire-wrapped fuel assemblies from PNC (Power reactor and Nuclear fuel development Corporation) and JNC (Japan Nuclear Cycle development institute). The research includes sensitivity studies on wall grid scales, GGI function, and turbulence models. The successful application of the RANS-based CFD methodology to the 217-pin wire-wrapped fuel bundles of KAERI PGSFR demonstrates its efficacy in practical scenarios, providing valuable insights for industrial management decisions.The studies conducted by K. ModenesiI, L. T. FurlanII, E. TomazI, R. GuirardelloI, and J. R. NúnezI (2004) have highlighted the future water shortage that humanity is expected to face. Consequently, it is crucial to develop innovative techniques and models to minimize the detrimental effects of pollution. Accurately predicting the environmental impact of new emissions in rivers, particularly during drought periods, is of utmost importance. Computational Fluid Dynamics (CFD) has emerged as a valuable tool for developing detailed models that analyze particle dispersion in rivers. However, due to the large number of points required to evaluate velocities and concentrations, these models necessitate powerful computational resources. In this context, the presented work introduces a new three-dimensional CFD-based model that is specifically designed for efficiency, significantly improving performance. The model's speed allows it to generate predictions for one-thousand-meter long river sections within a couple of hours using Pentium IV computers, while commercial CFD packages would take weeks to solve the same problem. Another innovative aspect of this work is the adoption of a half channel with a constant elliptical cross-section to represent the river, enabling the derivation of Navier-Stokes equations tailored to the elliptical system. Experimental data from the Atibaia River in São Paulo, Brazil, obtained from REPLAN (PETROBRAS refining unit), demonstrate good agreement with the model's predictions. The practical application of this research as a predictive tool can support and guide industrial management decisions.P. Amparo López-Jiménez et al. (2015) contribute to the growing utilization of CFD methods for characterizing hydrodynamics and mass transport in wastewater treatment units. While anaerobic digesters are a widely employed treatment method for stabilizing sludge before disposal, their hydrodynamics and mass transport characteristics have received comparatively less attention. The paper presents preliminary results from a 3D numerical study focusing on investigating the flow characteristics of sludge within the anaerobic digester at the Ontinyent Wastewater Treatment Plant in Valencia, Spain. The study adopts a Reynolds-Averaged Navier-Stokes (RANS) approach and utilizes the classical standard k-ε turbulence model for closure. Both Newtonian and Non-Newtonian behavior of sludge are considered using a single-phase model. The preliminary results highlight the presence of dead zones and potential shortcuts within the digester based on velocities and flow patterns. The analysis identifies sludge volumes within the digester that exhibit velocities lower than a predetermined settling velocity. The model is calibrated using available experimental pressure and temperature data. Additionally, suggestions and discussions are provided to mitigate dead zones, as well-mixed conditions are essential for effective anaerobic digestion.

Mehdi Ghadiri et al. (2015) present a novel model for simulating ion transport through porous media. The model builds upon the derivation of mass and momentum transfer equations for solutes in different phases, including feed, solvent, and membrane. Computational Fluid Dynamics (CFD) is employed to discretize the differential equations under steady-state conditions. The flow behavior in the extractor is modeled by numerically solving the Navier-Stokes equations using the finite element method. The simulation results demonstrate velocity profiles, pressure distribution, and solute concentration. Furthermore, the model's accuracy is validated by comparing the results of rubidium extraction with experimental data. The developed model provides insights into ion transport and contributes to the optimization of extraction processes.

Abbas Parsaie et al. (2015) address the complex task of analyzing the behavior and hydraulic characteristics of flow over dam spillways. Traditionally, physical and numerical methods have been employed for modeling such flows in water engineering projects. With advancements in Computational Fluid Dynamics (CFD) and the availability of high-performance computers, numerical methods have become more accessible and efficient for analyzing intricate flow phenomena. In this paper, the Flow 3D software is utilized to model the flow pattern at the guide wall of the Kamal-Saleh dam. The results reveal that the current geometry of the left wall causes flow pattern instability, leading to secondary and vortex flows in the initial approach channel. This shape of the guide wall negatively impacts the weir's performance in handling peak flood discharge. The study highlights the value of CFD in analyzing and optimizing hydraulic structures.ZhengrongQiang et al. (2018) focus on understanding particle velocity and movement in high-pressure abrasive water jet (AWJ) cutting processes, which play a crucial role in improving machining performance. The study employs a computational fluid dynamics (CFD) approach based on the Euler-Lagrange method and discrete particle modeling (DPM). The physical model considers the particle inlet position, particle inlet angle, and converge angle of the focus tube within the AWJ cutting head. The models are qualitatively and quantitatively validated using previous experimental data. The results demonstrate that a higher particle inlet position increases exiting velocities and reduces nozzle wear. Steeper abrasive feed angles improve particle acceleration and decrease radial velocity, resulting in reduced nozzle weight loss. The effects of the converging part angle are also analyzed. The findings contribute to enhancing machine efficiency, extending nozzle lifetime, and guiding the future design of AWJ nozzles.Mashallah Rezakazemi (2018) employs Computational Fluid Dynamics

(CFD) to model a counter-current flat-sheet membrane desalination system. The study aims to optimize the seawater purification process by investigating various system features and operating conditions. A 2D mathematical model based on the conservation equations for water molecules in different domains of the contactor module is presented. The finite element method (FEM) is employed to solve the governing equations. The model reveals that changes in operating conditions or process characteristics during seawater desalination can affect the overall system performance. Furthermore, the simulation and experimental results confirm the high efficiency of the contactor module in seawater purification.J. Bridgeman et al. (2010) utilize Computational Fluid Dynamics (CFD) to simulate turbulent flows in flocculation processes commonly employed in water treatment works. The paper explores various modeling strategies and simulation techniques, including steady and unsteady flow, different turbulence modeling approaches, and mesh density optimization. The analysis focuses on turbulence dissipation rate and demonstrates the benefits of using CFD in flocculation vessels for environmental engineering problem-solving. The research provides valuable insights into the development of models for improving water treatment processes.Maria Maza et al. (2015) introduce a three-dimensional numerical approach based on IHFOAM to study the interaction between tsunami waves and mangrove forests. The model considers solitary waves impinging on emergent rigid cylinders as an initial approximation. Two conceptual approaches are implemented using IHFOAM: solving the Unsteady Reynolds-Averaged Navier-Stokes (URANS) equations to directly simulate the flow field with the actual geometry of the cylinder array, and using a macroscopic approach with drag coefficient for wave forces calculation. The results demonstrate significant differences in the forces exerted on vegetation between uniform and random distributions. The study emphasizes the importance of considering realistic arrangements to accurately estimate wave-exerted forces on mangrove forests.

METHODOLOGY

Prerequisite for ANSYS Setup

1. Data had been collected from Tripura State Pollution Control Board (TSPCB) dated on 29/04/2016.

SI. No	Parameters	S-1	S-2	S-3	S-4	S-5	S-6	S-7	S-8	S-9	S-10
1	pH	7.57	7.56	7.2	7.31	7.08	7.06	6.86	7.21	7.01	6.84
2	DO	6.15	6.25	6.36	6.36	5.55	5.14	3.63	3.73	4.24	4.54
3	BOD	1.1	1.3	1.61	1.81	1	0.79	0.9	3.12	3.02	3.12
4	Fe	0.38	0.42	1.94	1.88	1.66	1.49	2..34	3.76	2.48	1.92
5	Cadmium	0.008	0.002	0.031	0.017	0.016	0.013	0.017	0.009	0.017	0.014
6	Phosphate	0.038	0.035	0.019	0.021	0.031	0.033	0.059	0.061	0.069	0.025

Courtesy TSPCB

Table1. Various positions of the Haorah River

LOCATIONS:

S-1. Confluence point of two streams (The point of origin)

S-2. Near National brick field, Jirania,

S-3. Near Ranirbazar market,

S-4. Near Chaturdash Devata bari bathing ghat, Khayerpur,

S- 5. Near bridge on Haorah River connecting Chandrapur and Baldakhal,

S-6. Near Aralia water intake point,

S-7. Near Bordowali water intake point,

S-8. Near Battala crematorium

S- 9. Dashamighat idol immersion side,

S-10. Near last point of Indian Territory entering Bangladesh.

♣ River section data is collected from earth pro by using time control of satellite imagery data dated on 14/05/2016.

♣ Satellite image is Geo-referenced for good results

Fig 1 Geo-referenced image of Haorah river section

ANALYSIS IN ANSYS FLUENT

Residuals:

When solver iterations are calculated, the residual sum of each conserved variable is computed and stored. This helps in recording the convergence history. In ideal process with infinite precision the residuals tend to be zero when the convergence occurs. However the scenario in actual computing is different. The residuals tend to be small valued and then stop changing. Their magnitudes differs with single precision to double precision ranging from six to twelve orders of magnitude respectively before rounding off. Errors: 2D geometry analysis on Fluent has certain errors on the pressure and the velocity contours. Since the geometry defines a boundary layer has some assumptions. 4.3 Mesh independence study The study is mesh independent i.e. solution that does not vary significantly even when the mesh is refined further. Details of Meshing from coarse to fine grid for section 1.

Trail 1:

Nodes	Elements	Max. Element size	Bounding box
377	213	0.002m	(0.12309*3.0194.e-02)mm

Velocity at outlet of section 1 is 0.5m/s

Trail 2:

Nodes	Elements	Max. Element size	Bounding box
9052	7556	0.0002m	(0.12309*3.0194.e-02)mm

Velocity at outlet of section 1 is 0.527m/s

Trail 3:

Nodes	Elements	Max. Element size	Bounding box
69976	65077	0.00004m	(0.12309*3.0194.e-02)mm

Velocity at outlet of section 1 is 0.534m/s

Trail 4:

Nodes	Elements	Max. Element size	Bounding box
316826	303500	0.00002m	(0.12309*3.0194.e-02)mm

Velocity at outlet section 1 is 0.534m/s

Table2. Meshing information

Therefore, grid independence of section 1 is at max. Element size of 0.00004m. Similarly, for section 2, section 3 & section 4 mesh independence study occur at max. Element size of 0.00004m.

♣ The input velocity at each section analysed is uniform averaged velocity because, the velocities are generally parabolic in nature which is not possible to given as a input. ♣ The important flow parameters like velocity, pressure and wall shear stress are plotted against the water quality parameters for the dependency.

♣ The modelled data is validated in MATLAB followed by ANN.

♣ The modelled data is compared with the experimental data of 15% error.

RESULT AND DISCUSSION

In this study 4 sections had simulated (the details of section are already discussed) During analysis Steady state is considered which is like the river flow condition.

Results of Section 1

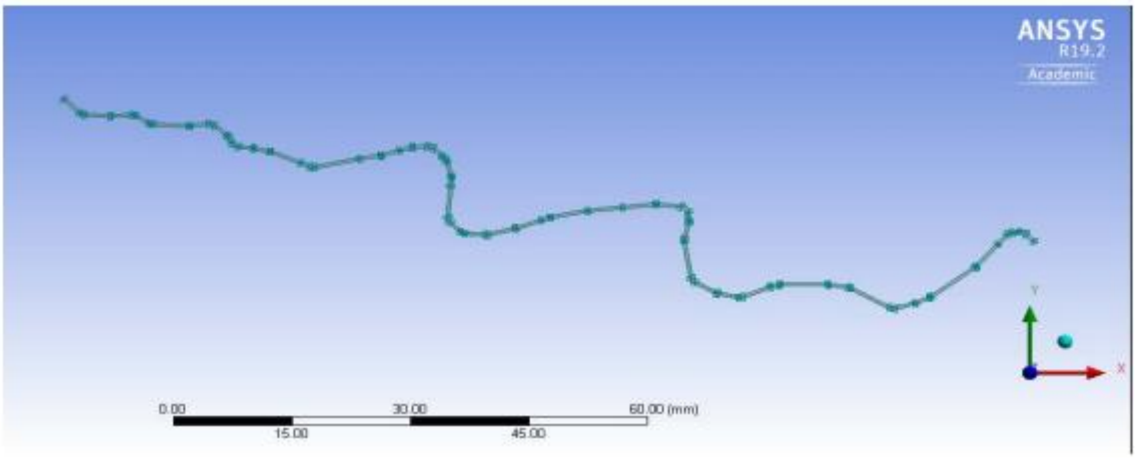

Figure 2. Geometry

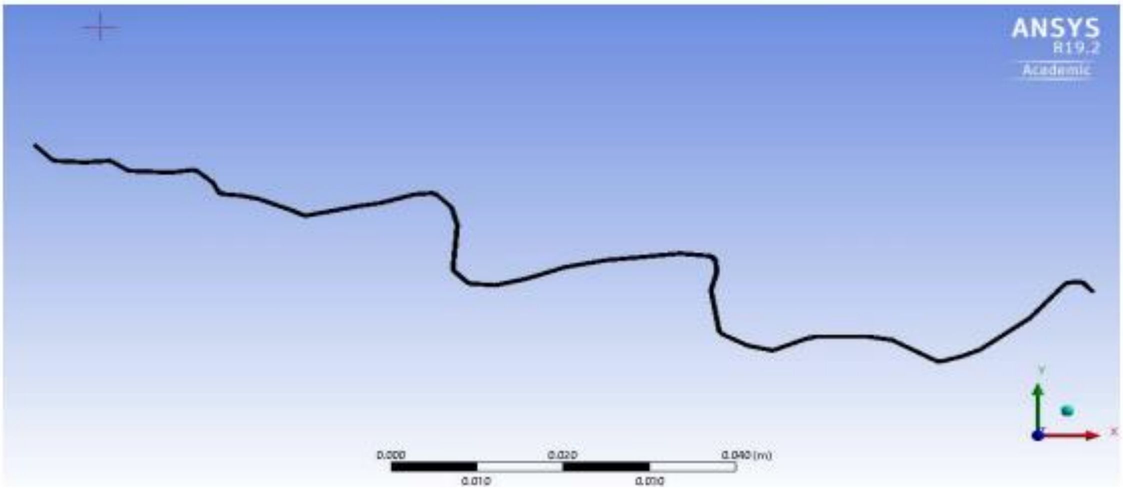

Figure 3. Meshing

Refined meshing of 0.00003m is made at the edges for accuracy results.

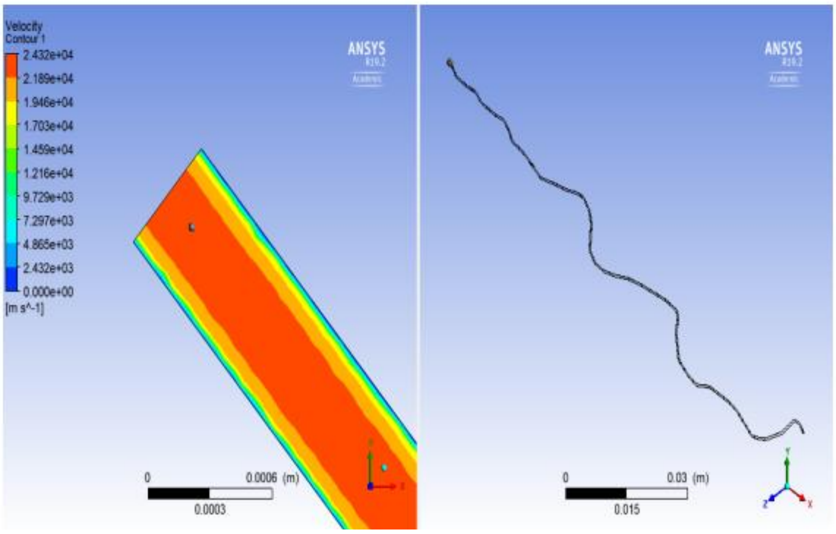

Figure 4. Velocity contours

As we can see from the above figure Velocity tends to decrease away from the centre and zero at the ends.

• For a given section at inlet the pressure is maximum, and velocity is minimum whereas at Outlet of the section the pressure is minimum, and velocity is maximum.

• In Fig. 4 wall shear is max at wall and zero at middle.

For all other sections also, we get the almost same kind of behaviours.

The scaling details of sections are

1)Section 1: 1:43030

2)Section 2: 1:43030

3)Section 3: 1:23790

4)Section 4: 1:9279

DATA VALIDATION IN MATLAB

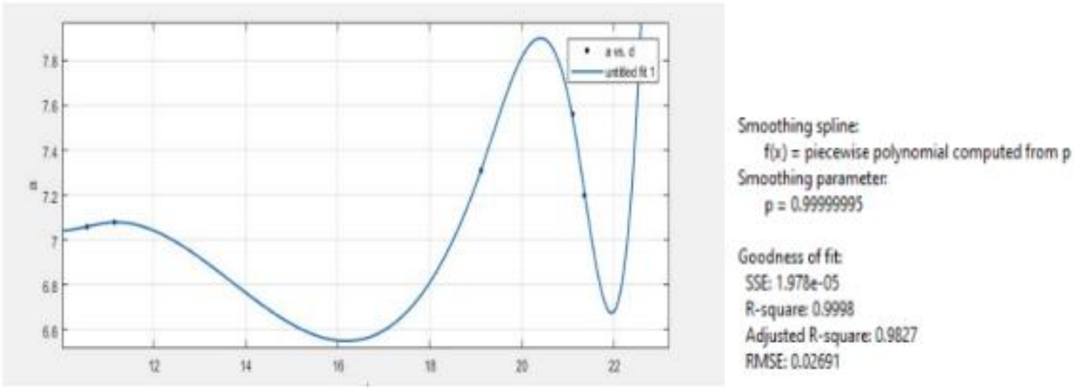

Figure 5. Graph of Velocity vs pH

The sample data validation in MATLAB is given here as velocity vs pH.The other validations were also done as pH vs Pressure, pH vs Shear stress, D.O vs Velocity, Graph D.O vs Pressure, D.O vs Shear stress, B.O.D vs Shear stress etc.

1. As discussed earlier for the whole study of river from Jirania to Aralia intake we have experimental data of WQP's at 5 points.
2. The above Fig. 5 gives the plot Velocity vs pH by using smoothing spline for fitting with regression value of 0.9998.

The other parameters as mentioned also fitted in MATLAB.

DATA VALIDATION IN ANN

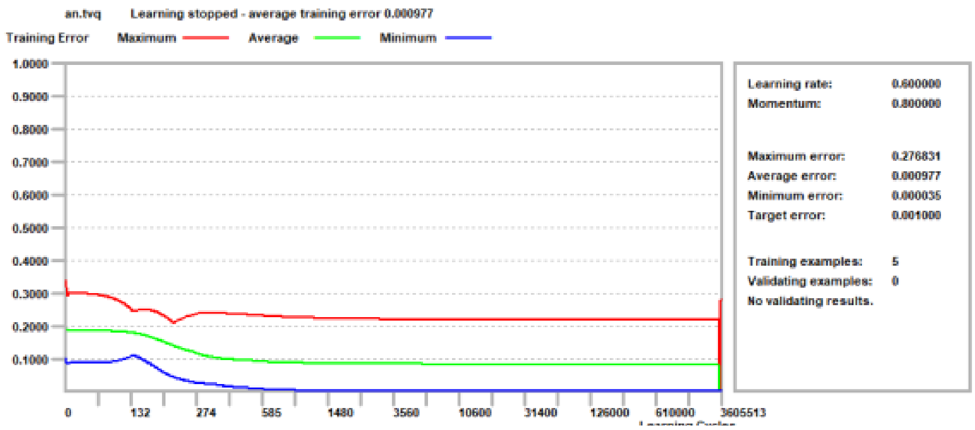

Figure 6. ANN Training Iterations for Velocity and Phosphate

Section Name	Flow Parameter		
	Velocity(m/s)	Pressure(Pa)	Wall Shear(Pa)
Section 1	0.5	14013.48	21.101
Section 2	0.534	17429.7	21.356
Section 3	0.57	15090.37	19.111
Section 4	0.6	728.82	11.146
at Outlet	0.636	6018.48	10.548

Table 3. Simulated Data

The other simulation results like velocity profile, shear stress, kinetic energy profiles were also found for all the stated sections and which can be produced on request.

parameters for validation	Experimental value	Validated value	Error (%)
pH vs Velocity	7.06	6.861	2.83
pH vs Pressure	7.06	7.887	11.7
pH vs Shear stress	7.06	7.258	2.8
DO vs Velocity	5.14	5.637	9.59
DO vs Pressure	5.14	5.587	8.85
DO vs Shear stress	5.14	5.55	4.71
BOD vs Shear stress	0.74	0.682	8.54
Fe vs Velocity	1.49	1.37	8.44
Cd vs Velocity	0.013	0.01225	6
Phosphate vs Velocity	0.033	0.0304	8.55
Phosphate vs Pressure	0.033	0.035	5.71
Phosphate vs Shear stress	0.033	0.0308	7.14

Table 4. Percentage of Error in Validated data

CONCLUSION

• The present study used various tools and software's like CFD ANSYS Fluent, AutoCAD, Earth Pro, MATLAB and ANN.
• With the help of CFD we tried to establish relationship between flow parameters and water quality parameters.
• Steady state condition is limited to the calmness of a river flow.
• As research modelling is 2D, limited parameters show the dependency and for the validation.
• The water quality parameters like pH and DO shows great certainty with given model.
• Very refined meshing is used to model the river sections, so that a good analysis can be performed.
• WQP's pH, Cd, DO, BOD, Fe and Phosphate shows dependency with the flow parameters.

Name	Flow Parameters		
	Velocity(m/s)	Pressure(Pa)	Wall Shear(Pa)
Section 1 inlet	0.5	14013.48	21.101
Section 2 inlet	0.534	17429.7	21.356
Section 3 inlet	0.57	15090.37	19.111
Section 4 inlet	0.6	728.82	11.146
Section 4 Outlet	0.636	6018.48	10.548

Table. 5. Values of Flow Parameters at different sections

REFERENCES

1. Amparo López-Jiménez, Juan Escudero-González, Tatiana Montoya Martínez, Vicente Fajardo Montañana, Carlo Gualtieri (2015), "Application of CFD methods to an anaerobic digester: The case of Ontinyent WWTP, Valencia, Spain", Journal of Water Process Engineering 7 (2015) 131-140, DOI: 10.1016/j.jwpe.2015.05.006.
2. Mehdi Ghadiri, , Mehdi Asadollahzadeh, Alireza Hemmati (2015), "CFD simulation for separation of ion from wastewater in a membrane contactor", Journal of Water Process Engineering Volume 6, June 2015, Pages 144-150.
3. Abbas Parsaie, Amir HamzehHaghiabi, Amir Moradinejad (2015), "CFD modeling of flow pattern in spillway's approach channel", Sustainable Water Resources Management September 2015, Volume 1, Issue 3, pp 245–251.
4. ZhengrongQiang, Meiping Wu, Xiaojin Miao, Rupy Sawhney (2018) "CFD research on particle movement and nozzle wear in the abrasive water jet cutting head", The International Journal of Advanced Manufacturing Technology April 2018, Volume 95, Issue 9–12, pp 4091–4100.
5. Mashallah Rezakazemi (2018), "CFD simulation of seawater purification using direct contact membrane desalination (DCMD)" system.Desalination, 2018, doi:10.1016/j.desal.2017.12.048
6. Bridgeman, J., et al. "The Development and Application of CFD Models for Water Treatment Flocculators." Advances in Engineering Software, vol. 41, no. 1, 2010, pp. 99– 109., doi:10.1016/j.advengsoft.2008.12.007.
7. Maria Maza, Javier L. Lara , Inigo J. Losadav (2015) Tsunami wave interaction with mangrove forests: A 3-D numerical approach, Coastal Engineering Volume 98, April 2015, Pages 33-54.
8. Ingham,,D.b.,and L. Ma. "Fundamental Equations for CFD in River Flow Simulations" Computational Fluid Dynamics, 2005, pp. 17–49., doi:10.1002/0470015195.ch2.
9. Modenesi, K., et al. "A CFD Model for Pollutant Dispersion in Rivers." Brazilian Journal of Chemical Engineering, vol. 21, no. 4, 2004, pp. 557–568., doi:10.1590/s0104- 66322004000400005.
10. Khaldi, Nawel, et al. "Distribution Characteristics of Pollutant Transport in a Turbulent Two-Phase Flow." Environmental Science and Pollution Research, vol. 22, no. 8, 2015, pp. 6349– 6358., doi:10.1007/s11356-014-3664-3.

11. Bradbrook, K.F.; Lane, S.N. and Richards, K.S.,(2010) "Numerical Simulation of Threedimensional Time-Averaged Flow Structure at River Channel Confluences". Water Resource. Res. 36 (9), 2731-2746, 2000.
12. Lin Ma;, Ashworth, P.J.; Best, J.L.; Elliott, L.; Ingham, D.B. and Whitecombe,L.J. "Computational Fluid Dynamics and the Physical Modeling of an Upland Urban River Geomorphology",44, 375-391, 2002.
13. M. Fasihi, S. Shirazian, A. Marjani, M. Rezakazemi, (2018),"Computational fluid dynamics simulation of transport phenomena in ceramic membranes for SO2 separation", Math. Computational. Model. 56 (2012) 278–286.
14. Tabbara M, Chatila J, Awwad R (2005), "Computational simulation of flow over stepped spillways", Comput Struct 83:2215–2224.
15. S. Shirazian, M. Rezakazemi, A. Marjani, M.S. Rafivahid, "Development of a mass transfer model for simulation of sulfur dioxide removal in ceramic membrane contactors", Asia Pac. J. Chem. Eng. 7 (2012) 828–834.
16. M. Rezakazemi, M. Iravaninia, S. Shirazian, T. Mohammadi, "Transient computational fluid dynamics modeling of pervaporation separation of aromatic/aliphatic hydrocarbon mixtures using polymer composite membrane", Polym. Eng. Sci. 53 (2013) 1494–1501.
17. Y. Kawahara. "Modeling of free surface effects on turbulent flow in open channel Proc."(2001) Meeting of JSFM., 59-60 (in Japanese), 2001.
18. K. Onitsuka and I. Nezu. "Numerical prediction of rectangular open-channel flow by using large eddy simulation". 29th IAHR Congress Proc., Theme D., 1:196-201, 2001.
19. A. Nakayama and S. Yokojima. "LES of openchannel flow with free-surface fluctuation. Proceedings of Hydraulic Engineering", JSCE, 46:373-378, 2002.
20. K. Onitsuka and I. Nezu. "Numerical prediction of rectangular open-channel flow by using large eddy simulation". 29th IAHR Congress Proc., Theme D., 1:196-201, 2001.
21. Alhassan H. Ismail (2018) "Estimation of river Tigris dispersivities using a steady-state numerical model", Applied Water Science 8:108.
22. Maria Maza, Javier L. Lara , Inigo J. Losadav (2015) "Tsunami wave interaction with mangrove forests: A 3-D numerical approach", Coastal Engineering Volume 98, April 2015, Pages 33-54.
23. Abbas Parsaie, Amir HamzehHaghiabi, Amir Moradinejad (2015) "CFD modeling of flow pattern in spillway's approach channel", Sustainable Water Resources Management September 2015, Volume 1, Issue 3, pp 245–251.