9TH International Symposium on HYDRAULIC STRUCTURES

24-27 October, 2022
IIT ROORKEE, INDIA

ISHS 2022 HANDBOOK

IAHR AIRH
International Association for Hydro-Environment Engineering and Research

Hosted by
Spain Water and IWHR, China

भारतीय प्रौद्योगिकी संस्थान रुड़की
INDIAN INSTITUTE OF TECHNOLOGY ROORKEE

175 YEARS OF IIT ROORKEE
Estd. 1847

CONTENTS

LETTER FROM THE CHAIRMAN

On behalf of the Local Organising Committee, I welcome all the delegates of this 9th International Symposium on Hydraulic Structures at the Department of Civil Engineering, IIT Roorkee, India. I also welcome you all to this Institute which has a long history of its existence. Established in 1847 by the British Empire as Roorkee College, primarily to help the construction of Ganga Canal under the directions of Col. Proby Cautley. This canal was conceived after the severe famine occurred in North India during 1837-38. This College was re-named as Thomason College of Civil Engineering in 1854 and then became the first Engineering University of Independent India as University of Roorkee in 1949 and finally as IIT Roorkee in 2001.

Roorkee is a city in North India and spread over a flat terrain under Shivalik Hills of Himalayas. In addition to many Departments of IIT Roorkee, two well-known Institutes i.e., National Institute of Hydrology and Irrigation Research Institute, work in the field of Hydrology, Hydraulics and Water Resources. The Upper Ganga Canal, passing through the heart of the Roorkee City, beautifies Roorkee many folds. Weather of Roorkee is pleasant in the month of the October - the first month of the post-monsoon - having temperatures varying from 18 to 30 degrees Celsius.

With the frequent floods, droughts and acute temporal & spatial variations in the water resources, it is very important to store, divert and regulate the flow by constructing necessary hydraulic structures. Recent advancements, in the analysis and design of the hydraulic structures, have helped in an economical, eco-friendly and sustainable structures. The present symposium is in series of similar earlier successful events at Aachen (ISHS2018), Portland (ISHS2016), Brisbane (ISHS2014), Porto, Portugal (ISHS2012) etc. I am sure the present symposium will provide a distinctive opportunity for the engineers and researchers working in this area to present their works and to share the recent advancements in the field of hydraulic structures.

We originally received 115 abstracts and after final paper submission and a thorough peer-review process, 60 research papers were accepted for the presentation and inclusion in the Proceedings.

This final brochure provides details of Technical Programs, Keynote Speakers, Technical Tours, Workshops, Various Committees and Abstracts of 60 accepted papers. Accepted and presented papers will be allocated a direct object identifier (DOI) and published in the proceedings of the Symposium, that will be made freely available at the Utah State University Digital Commons. Extended versions of the selected papers shall be published in a special edition of Journal of Hydraulic Engineering, ASCE.

Once again, I welcome you all in this ISHS 2022 at IIT Roorkee!

Zulfequar Ahmad
Conference Chair

ABOUT IIT ROORKEE

Indian Institute of Technology Roorkee is among the foremost institutes of national importance in higher technological education and in engineering, basic and applied research. Since its establishment, the Institute has played a vital role in providing the technical manpower. The Institute ranks amongst the best technological institutions in the world and has contributed to all sectors of technological development. It has also been considered a trend-setter in the area of education and research in the field of science, technology, and engineering. The Institute had celebrated its Sesquicentennial in October 1996 and is now celebrating 175 years of its existence. The Institute offers Bachelor's Degree courses in 10 disciplines of Engineering and Architecture and Postgraduate's Degree in 55 disciplines of Engineering, Applied Science, Architecture and planning. The Institute also has the facility for doctoral work in all Departments and Research Centres.

The Department of Civil Engineering at the Indian Institute of Technology Roorkee is the oldest and the largest in the country and is considered as the best in the country for the education in Civil Engineering. It was established on October 19, 1847 as Roorkee College of Civil Engineering and renamed as the Thomason College of Civil Engineering in 1854. The department is producing several eminent engineers who are making notable contributions in the planning and execution of Civil Engineering projects in India as well as abroad.

The Department offers a four-year course leading to the Bachelor's Degree in Civil Engineering and two-year courses leading to Master's degree in six major specialisations of civil engineering (viz., Environmental Engineering, Geomatics Engineering, Geotechnical Engineering, Hydraulics Engineering, Structural Engineering and Transportation Engineering). These programmes are supported with strong doctoral programmes in all the specialisations. The faculty of the department continues to strive loftier by exploring new frontiers of knowledge, imparting the latest technical knowledge to the students and conducting high quality of research

Hydraulics Laboratory of Civil Engineering Department was established in 1956, having a floor area of 4400 m2 and a discharge of 1.0 m3/s. The laboratory has facilities for field and laboratory testing such as: Tilting/Non-tilting flumes, Open/Closed circuit wind tunnel, River model tray, Outdoor River and hydraulic model space. The laboratory is equipped with state-of-the-art equipment such as PIV, ADV, mini LDV, Micro S, Scour monitor, Bed profiler, Ultrasonic/Magnetic flowmeters, Sediment concentration meter, etc.

ORGANIZATION

A. International Scientific Committee

1. Michele Palermo (Chair), University of Pisa, Italy
2. Abdorreza Kabiri-Samani, Isfahan University of Technology, Iran
3. Ajay K. Singh, NTPC, India
4. Ajay Pradhan, Cetus Consulting Solution Services Private Limited, India
5. Akbar Safarzadeh, University of Mohaghegh Ardabili, Iran
6. Alessandro Valiani, University of Ferrara, Italy
7. Anton J. Schleiss, EPFL, Switzerland
8. Arturo Marcano, CVG EDELCA, Venezuela
9. Arun Kumar, IIT Roorkee, India
10. Banjamin Dewals, Liege University, Belgium
11. Benjamin Hohermuth, ETH Zurich, Switzerland
12. Blake Tullis, Utah State University, USA
13. Brian Crookston, Utah State University, USA
14. Brown Jesse, U.S. Army Corps of Engineers, USA
15. Bruce Melville, Auckland University, New Zealand
16. Cabrera Juan, Universidad Nacional de Ingeneria, Portugal
17. Cecilia Lopardo, Istituto National del Agua, Argentina
18. Chandrashekhar Ojha, IIT Roorkee, India
19. Claudio Fattor, Istituto National del Agua, Argentina
20. Cristián Escauriaza, Universidad Católica de Chile, Chile
21. Daniel B. Bung, Aachen University of Applied Sciences, Germany
22. Daniel Valero Huerta, IHE Delft, The Netherlands
23. Deep Roy, University of Pisa, Italy
24. Dipankar Roy, Madanapalle Institute of Technology and Science, India
25. Fabian Bombardelli, University of California, Davis, USA
26. Feimster Laura, Schnabel Engineering, USA
27. Francesco Granata, Università degli studi di Cassino e del Lazio Meridionale, Italy
28. Giuseppe Oliveto, Università degli studi della Basilicata, Italy
29. Hazi Azamathulla, University of the West Indies, Trinidad
30. Hotchkiss Rollin, Brigham Young University, USA
31. Hubert Chanson, University of Queensland, Australia
32. Iacopo Carnacina, Liverpool John Moores University, UK
33. Isabella Schalko, ETH Zurich, USA
34. James Yang, Royal Institute of Technology, Sweden
35. Jorge Matos, University of Lisbon, Portugal
36. Jorge D. Bacchiega, Istituto National del Agua, Argentina
37. Jose M. Adriasola, Betchel, Chile
38. José M. Carrillo, Universidad Politecnica de Cartagena, Spain
39. Juan Pablo Toro Labbe, Universidad Andres Bello, Chile
40. Khodier Mohanad, Yarmouk University, Jordan
41. Kramer Matthias, UNSW Canberra, Australia
42. KS Hariprasad, IIT Roorkee, India

43. Lesleighter Eric, Lesleighter Consulting, Australia
44. Luis G. Castillo, Universidad Politecnica de Cartagena, Spain
45. Mario Oertel, Helmut-Schmidt-University Hamburg, Germany
46. Masoud Ghodsian, Tarbiat Modares University, Iran
47. Mazurek Kerry, University of Saskatchewan, Canada
48. Michael Pfister, University of Applied Sciences and Arts Western Switzerland HES-SO, Switzerland
49. Moran Rafael, Universidad Politecnica de Madrid, Spain
50. Mortensen Joshua, U.S. Bureau of Reclamation, USA
51. Narendra K. Goel, IIT Roorkee, India
52. Neff Keil, Stantec, USA
53. Nils Ruther, NTNU, Norway
54. Óscar Link, Universidad de Concepción, Chile
55. Pablo Spalletti, Istituto National del Agua, Argentina
56. Patricio A. Moreno-Casas, University of the Andes, Chile
57. Phillips Michael, U.S. Army Corps of Engineers, USA
58. Pramod K. Sharma, IIT Roorkee, India
59. Prem L. Patel, SVNIT Surat, India
60. Rajib Das, Jadavpur University, India
61. Rita Carvalho, University of Coimbra, Portugal
62. Robert Boes, ETH Zurich, Switzerland
63. Robert Janssen, Betchel, Australia
64. Roshni Thendiyath, National Institute of Technology Patna, India
65. Ruswick Kevin, Schnabel Engineering, USA
66. Sahameddin Mahmoudi Kurdistani, IA.ING – Hydraulic Division, Italy
67. Sanjay Giri, Deltares, The Netherlands
68. Sean Mulligan, National University of Ireland, Galway, Ireland
69. Sebastien Erpicum, Liege University, Belgium
70. Seyed Haji, South Florida Water Management District, USA
71. Silke Wieprecht, University of Stuttgart, Germany
72. Silvia Meniconi, Università degli Studi di Perugia, Italy
73. Simone Pagliara , ETH Zurich, Switzerland
74. SK Zeeshan Ali, IIT Hyderabad, India
75. Stefan Felder, University of New South Wales, Australia
76. Stefano Pagliara, Universty of Pisa, Italy
77. Subhasish Das, Jadavpur University, India
78. Subhasish Dey, IIT Kharagpur, India
79. Teal Marty, West Consultants, USA
80. Thanos Papanicolaou, Northwest hydraulic consultants, USA
81. Valentin Heller, The University of Nottingham, UK
82. Vito Ferro, University of Palermo, Italy
83. Wahl Tony, U.S. Bureau of Reclamation, USA
84. Youichi Yasuda, Nihon University, Japan
85. Zhu David, University of Alberta, Canada
86. Zulfequar Ahmad, IIT Roorkee, India

B. Local Organizing Committee

1. Zulfequar Ahmad, IIT Roorkee (Conference Chair)

2. Ajay K Singh, THDC, Rishikesh, India
3. Ajay Pradhan, C2S2 Pvt. Ltd., India
4. Arun Kumar, IIT Roorkee, India
5. Ashish Pandey, IIT Roorkee, India
6. C S P Ojha, IIT Roorkee, India
7. KS Hariprasad, IIT Roorkee, India
8. N K Goel, IIT Roorkee, India
9. P K Sharma, IIT Roorkee, India
10. R Vinnarasai, IIT Roorkee, India

C. Editorial Board

1. Michele Palermo (Scientific Committee Chair), University of Pisa, Italy
2. Zulfequar Ahmad (Local Organizing Committee Chair), IIT Roorkee, India
3. Brian Crookston (IAHR Committee on Hydraulic Structures Chair), Utah State University, USA
4. Sebastien Erpicum (IAHR Committee on Hydraulic Structures Past Chair), Liege University, Belgium

D. Reviewers

1. Abdorreza Kabiri-Samani, Isfahan University of Technology, Iran
2. Ajay K. Singh, NTPC, India
3. Ajay Pradhan, Cetus Consulting Solution Services Private Limited, India
4. Alessandro Valiani, University of Ferrara, Italy
5. Anton J. Schleiss, EPFL, Switzerland
6. Arturo Marcano, CVG EDELCA, Venezuela
7. Banjamin Dewals, Liege University, Belgium
8. Benjamin Hohermuth, ETH Zurich, Switzerland
9. Blake Tullis, Utah State University, USA
10. Brian Crookston, Utah State University, USA
11. Bruce Melville, Auckland University, New Zealand
12. Cecilia Lopardo, Istituto National del Agua, Argentina
13. Chandrashekhar Ojha, IIT Roorkee, India
14. Claudio Fattor, Istituto National del Agua, Argentina
15. Daniel B. Bung, Aachen University of Applied Sciences, Germany
16. Daniel Valero Huerta, IHE Delft, The Netherlands
17. David Zhu, University of Alberta, Canada
18. Deep Roy, University of Pisa, Italy
19. Dipankar Roy, Madanapalle Institute of Technology and Science, India
20. Erik Lesleighter, Lesleighter Consulting, Australia
21. Fabian Bombardelli, University of California, Davis, USA
22. Francesco Granata, Università degli studi di Cassino e del Lazio Meridionale, Italy
23. Giuseppe Oliveto, Università degli studi della Basilicata, Italy
24. Hazi Azamathulla, University of the West Indies, Trinidad
25. Hubert Chanson, University of Queensland, Australia
26. Iacopo Carnacina, Liverpool John Moores University, UK
27. Isabella Schalko, ETH Zurich, USA
28. James Yang, Royal Institute of Technology, Sweden

29. Jesse Brown, U.S. Army Corps of Engineers, USA
30. Jessica Di Nardi, University of Pisa, Italy
31. Jorge Matos, University of Lisbon, Portugal
32. Jorge D. Bacchiega, Istituto National del Agua, Argentina
33. Jose M. Adriasola, Betchel, Chile
34. José M. Carrillo, Universidad Politecnica de Cartagena, Spain
35. Juan Cabrera, Universidad Nacional de Ingeneria, Portugal
36. Juan Pablo Toro Labbe, Universidad Andres Bello, Chile
37. Keil Neff, Stantec, USA
38. Kerry Mazurek, University of Saskatchewan, Canada
39. Kevin Ruswick, Schnabel Engineering, USA
40. KS Hariprasad, IIT Roorkee, India
41. Laura Feimster, Schnabel Engineering, USA
42. Luis G. Castillo, Universidad Politecnica de Cartagena, Spain
43. Mario Oertel, Helmut-Schmidt-University Hamburg, Germany
44. Marty Teal, West Consultants, USA
45. Masoud Ghodsian, Tarbiat Modares University, Iran
46. Mathias Kramer, UNSW Canberra, Australia
47. Michael Phillips, U.S. Army Corps of Engineers, USA
48. Michael Pfister, University of Applied Sciences and Arts Western Switzerland HES-SO, Switzerland
49. Michele Palermo, University of Pisa, Italy
50. Mohanad Khodier, Yarmouk University, Jordan
51. Narendra K. Goel, IIT Roorkee, India
52. Nils Ruther, NTNU, Norway
53. Óscar Link, Universidad de Concepción, Chile
54. Pablo Spalletti, Istituto National del Agua, Argentina
55. Patricio A. Moreno-Casas, University of the Andes, Chile
56. Pramod K. Sharma, IIT Roorkee, India
57. Rafael Moran, Universidad Politecnica de Madrid, Spain
58. Rajib Das, Jadavpur University, India
59. Rita Carvalho, University of Coimbra, Portugal
60. Robert Boes, ETH Zurich, Switzerland
61. Robert Mohanand. Betchel, Australia
62. Roshni Thendiyath, National Institute of Technology Patna, India
63. Sahameddin Mahmoudi Kurdistani, IA.ING – Hydraulic Division, Italy
64. Sanjay Giri, Deltares, The Netherlands
65. Sean Mulligan, National University of Ireland, Galway, Ireland
66. Sebastien Erpicum, Liege University, Belgium
67. Silke Wieprecht, University of Stuttgart, Germany
68. Silvia Meniconi, Università degli Studi di Perugia, Italy
69. Simone Pagliara , ETH Zurich, Switzerland
70. SK Zeeshan Ali, IIT Hyderabad, India
71. Stefan Felder, University of New South Wales, Australia
72. Stefano Pagliara, Universty of Pisa, Italy
73. Subhasish Das, Jadavpur University, India
74. Subhasish Dey, IIT Kharagpur, India
75. Thanos Papanicolaou, Northwest hydraulic consultants, USA

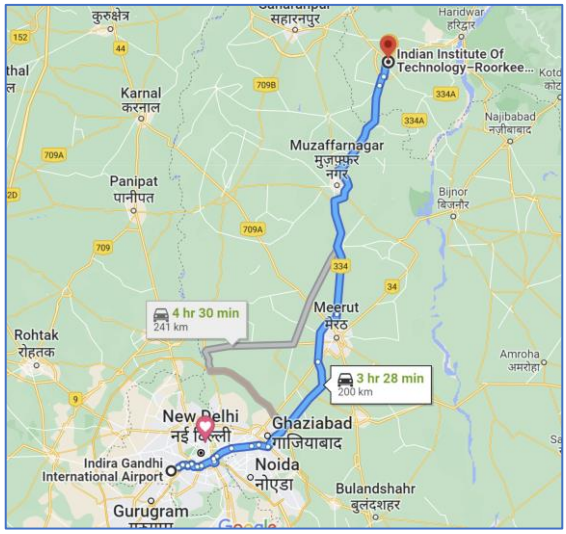
76. Tony Wahl, U.S. Bureau of Reclamation, USA
77. Valentin Heller, The University of Nottingham, UK
78. Vito Ferro, University of Palermo, Italy
79. Youichi Yasuda, Nihon University, Japan
80. Zulfequar Ahmad, IIT Roorkee, India

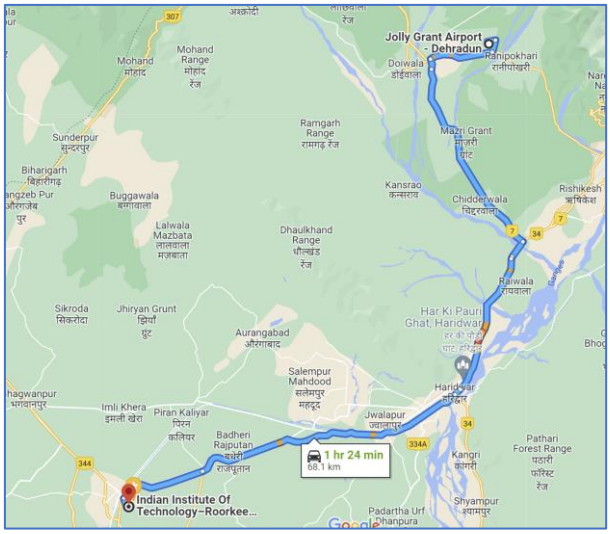

Bokaro Barrage, India


Tilaiya dam, India

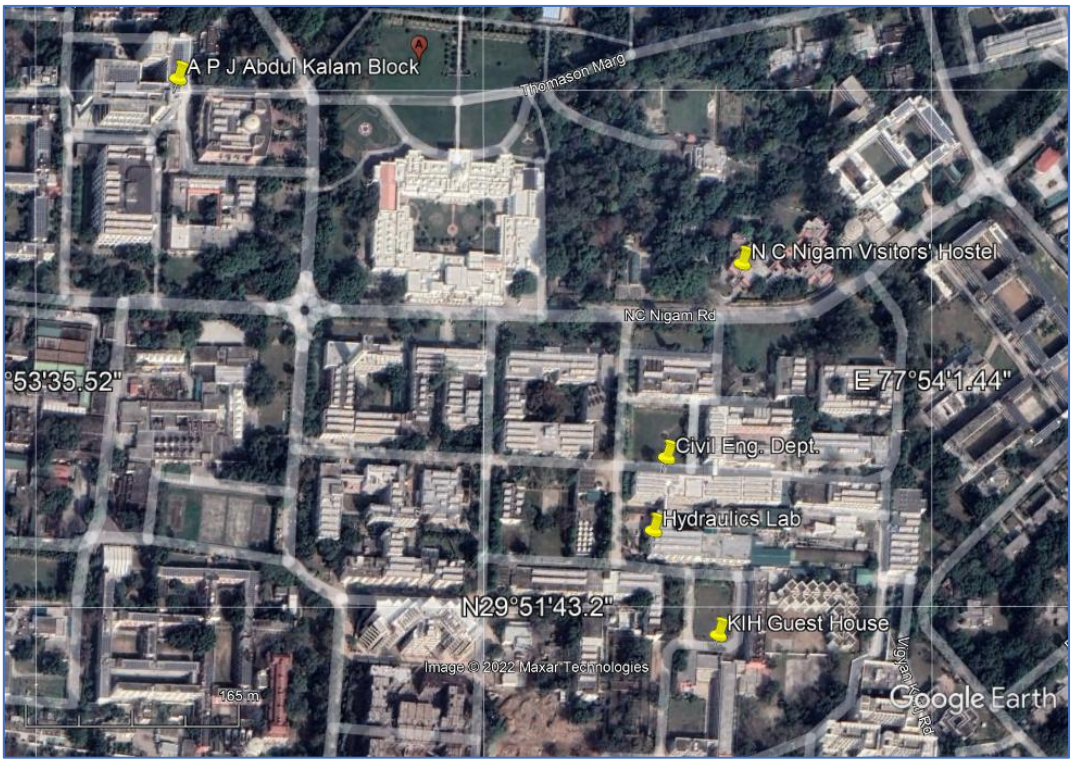
VENUE

Roorkee is a city in North India and spread over a flat terrain under Sivalik Hills of Himalayas. The city is developed on the banks of Ganges Canal, its dominant feature, which flows from north–south through middle of the city. It is well connected with New Delhi, the capital of India by train (https://etrain.info/in) and road. Two preferred trains that run between New Delhi and Roorkee are Shatabdi Express and Jan Shatabdi Express. Nearest domestic airport is about 65 km from Roorkee at Dehradun while International airport is Indira Gandhi International (IGI) Airport, New Delhi, which is about 200 km from Roorkee. A cab takes about four hours to travel from IGI airport to Roorkee and about one and half hour from Dehradun Aiport to Roorkee. The railway system and roadways bus services offer convenient and inexpensive transportation to nearby areas of tourist attraction like Haridwar, Rishikesh, Kaliar sharif etc. Also, there is a wide variety of appealing restaurants, city and outdoor activities, and tourist attractions available for attendees to enjoy.





The ISHS2022 shall be held at Department of Civil Engineering (29° 51' 46.59 " N, 77° 53' 53.64" E) and A P J Abdul Kalam Block (29° 51' 56.10" N, 77° 53' 39.53 " E) of the IIT Roorkee. Their locations are shown below:


A P J Abdul Kalam Block
Thomason Marg
N C Nigam Visitors' Hostel
N.C. Nigam Rd
°53'35.52"
E 77°54'1.44"
Civil Eng. Dept.
Hydraulics Lab
N29°51'43.2"
KIH Guest House
Image © 2022 Maxar Technologies
165 m
Google Earth


Civil Engineering Department


A P J Abdul Kalam Block

Accommodation of the delegates are arranged in the Guest Houses of the IIT Roorkee i.e., N C Nigam Visitor's Hostel (29° 51' 51.45" N, 77° 53' 55.83" E) and Khosla International House (29° 51' 42.14" N, 77° 53' 55.13" E).

TECHNICAL PROGRAM

A. Program at a Glance

October 24, 2022		
Time	**Activity**	
08:00 - 13:00	Technical Tour-I	Upper Ganga Canal
October 25, 2022		
Time	**Activity**	
10:00 -13:00	Workshop on Non-linear Weirs	Conference Room, Civil Eng. Department
13:00 - 14:00	Lunch	Lawn of Civil Eng. Department
14:00 –16:00	Workshop on Non-linear Weirs	Conference Room, Civil Eng. Department
16:00 – 16:30	Tea Break	Lawn of Civil Eng. Department
16:30 – 17:30	Hands-on Experiments	Hydraulics Laboratory, Civil Eng. Department
19:30 - 21:30	Workshop Dinner	Lawn of Civil Eng. Department

October 26, 2022			
Time	**Activity**		
08:00 - 09:30	Registration	Room No. L2-005 A P J Abdul Kalam Block	
09:30 - 10:30	Inauguration	Room No. L2-103 A P J Abdul Kalam Block	
10:30 - 11:00	High Tea	A P J Abdul Kalam Block	
11:00 – 11:40	Keynote Lecture-1	**Fluid mechanics past hydraulic structures: A shift in paradigm** Fabian A. Bombardelli, University of California, Davis L2-103	
11:40 - 12:20	Keynote Lecture-2	**Experiences from training the Brahmaputra-Jamuna River** Erik Mosselman, Deltares, The Netherlands L2-103	
12:20 - 13:00	Keynote Lecture-3	**The Challenge of Rock Scour Assessment at High Head Spillways,** Anton J. Schleiss, EPFL, Switzerland L2-103	
13:00 - 14:00	Lunch	A P J Abdul Kalam Block	
14:00 - 15:30	Technical Sessions	TS-1: Spillways L2-103	TS-2: Energy Dissipators L2-104
15:30 - 16:00	Tea Break	A P J Abdul Kalam Block	
16:00 - 17:30	Technical Sessions	TS-3: Intakes and outlets L2-103	TS-4: Energy Dissipators L2-104
17:30 - 18:30	Open HS Committee meeting	L2-103	
19:30 - 22:30	Symposium Dinner	Community Centre	

October 27, 2022				
Time	**Activity**			
09:00– 09:40	Keynote Lecture-4	**Modifying the Mosul Dam Bottom Outlet and Flip Bucket** Mike Phillips, US Army Corps of Engineers L2-103		
09:40 – 10:20	Keynote Lecture-5	**Composite Modeling for Hydraulic Structures Design** Sebastien Erpicum, Liege University, Belgium L2-103		
10:20 – 11:00	Keynote Lecture-6	**Hydraulic Behaviour of In-Stream Low Head Structures for River Restoration,** Stefano Pagliara, University of Pisa, Italy L2-103		
11:00 - 11:30	Tea Break	A P J Abdul Kalam Block		
11:30 - 13:00	Technical Sessions	TS-5:Non-Linear Weirs L2-103	TS-6: Sediment Transport L2-104	
13:00 - 14:00	Lunch	A P J Abdul Kalam Block		
14:00 - 15:30	Technical Sessions	TS-7: Physical & Numerical Modeling L2-103	TS-8: Scour around Hydraulic Structures L2-104	TS-9: Training and Protection Works L2-005
15:30 - 16:00	Tea Break	A P J Abdul Kalam Block		
16:00 - 17:30	Technical Sessions	TS-10: Physical and Numerical Modeling L2-103	TS-11: Scour around Hydraulic Structures L2-104	TS-12: Miscellaneous L2-005
17:30 - 18:00	Valedictory Function	L2-103 Phil H. Burgi Best Paper Award Announcement		
18:00 - 18:30	Valedictory Tea	A P J Abdul Kalam Block		
19:30 - 22:00	Dinner	Lawn of Civil Eng. Dept.		
October 28, 2022				
Time	**Activity**			
08:00 - 19:00	Technical Tour-II	Tehri Dam		


Konar dam, India

B. Technical Sessions

Technical Session: TS-1	Date : 26/10/2022	Time: 1400 to 1530 Hrs
Theme: Spillways	Location: L2-103, A P J Abdul Kalam Block	

S No.	Paper ID	Title	Author(s)
1	1432	Surface Velocities and Free-surface Aeration in a Converging Smooth Chute during a Major Flood Event	Hubert Chanson, C.J. Apelt
2	1480	Experimental and Numerical Investigation on Critical Submergence for Square Water Intakes	Bhagwan Das, Z. Ahmad, Pramod Kumar Sharma
3	1442	Hydraulic Behavior of Spill channel of Polavaram dam spillway without left embankment	Vankayalapati S. Ramarao, Yogendra Nath Srivastava
4	1479	Recent challenges in design of spillway – An Indian scenario	R.R. Bhate
5	1435	Non-intrusive measurements of air-water flow properties in supercritical flows down grass-lined spillways	H. Cui, S, Felder, M. Kramer

Technical Session: TS-2	Date : 26/10/2022	Time: 1400 to 1530 Hrs
Theme: Energy Dissipators	Location: L2-104, A P J Abdul Kalam Block	

S No.	Paper ID	Title	Author(s)
1	1446	Model Studies for Desilting Basin for Teesta-VI H.E. Project, Sikkim – A case study	M.Z. Qamar, M.K. Verma, A.P. Meshram, Neena Isaac
2	1472	Hydrodynamic behavior of stilling basins with deflectors to reduce supersaturation of total dissolved gases: analysis of Yacyretá and Jorge Cepernic dams	C.A. Fattor, J.D. Bacchiega, F. Romero, C. Fernández
3	1448	Numerical and Experimental Study on Energy Dissipation in Hydraulic Jump: A Comparison between Horizontal and Sloping Rough Channel Bed	Koroungamba Laishram, Thiyam Tamphasana Devi, Ngangbam Romeji Singh, Potsangbam Albino Kumar
4	1496	Investigation of Energy Attenuation, Flow Resistance And Impending Motion Of Downstream Bed Material In Block Ramps	Vishal Singh Rawat, Roshni Thendiyath, Michele Palermo, Simone Pagliara, Deep Roy
5	1431	Energy dissipation and flow regime downstream of trapezoidal piano key weirs	Xiaoyang Shen, Mario Oertel

Technical Session: TS-3	Date : 26/10/2022	Time: 1600 to 1730 Hrs
Theme: Intakes and outlets	Location: L2-103, A P J Abdul Kalam Block	

S. No.	Paper ID	Title	Author(s)
1	1459	Feasibility Assessment of the Application of Delft3D Model to Simulate Flow over a Spillway in a Dam: Study Case of Hampaturi Dam in Bolivia	F.D. Oliveira, M.W. Heredia
2	1478	Antivortex device to operate pump intakes below the minimum submergence	Mauricio Abel Angulo, Arturo Rivetti, Cecilia Verónica Lucino, Sergio Oscar Liscia
3	1445	A Novel Dual Inlet Tangential Vortex Drop Shaft: An Analytical and Numerical Study	Sean Mulligan, Shane Pope
4	1495	Submerged flow regimes downstream of a weir at multiple slopes	K.S. Vikas, A. Sarkar
5	1437	Prediction of Discharge Coefficient of Circular Side Orifice through Machine Learning Technique	Syed Ahmed Noman, Ajmal Hussain, Mohd. Danish, Rashid Ali

Technical Session: TS-4	Date : 26/10/2022	Time: 1600 to 1730 Hrs
Theme: Energy Dissipators	Location: L2-104, A P J Abdul Kalam Block	

S No.	Paper ID	Title	Author(s)
1	1440	Effect of deficient tailwater on the performance of slotted roller bucket of Indira Sagar Dam, Madhya Pradesh	Vankayalapati S. Ramarao, Amit Kulhare
2	1473	Angle Effects of LIDAR Measurements on a Flat Surface and in High-velocity Spillway Flows	R. Li , K.D. Splinter M. Kramer & S. Felder
3	1427	Preliminary Evaluation and Design of a New Energy Dissipation Stilling Basin via Numerical and Experimental Modeling	F. Zabaleta, F.A. Bombardelli, S.A. Valbuena
4	1456	Stilling Basins Using Wedge-shaped Baffle Blocks	N. N. Pillai, M.L. Kansal
5	1469	Physical and Numerical Model Studies for Hydraulic Design of Stilling Basin as an Energy Dissipator of a Spillway-A Case Study	B.S. Sundarlal, Vaishali Prakash Gadhe, S.R. Patnaik, Amol Hanmantrao Gaikwad

Technical Session: TS-5	Date : 27/10/2022	Time: 1130 to 1300 Hrs
Theme: Non-linear weirs	Location: L2-103, A P J Abdul Kalam Block	

S No.	Paper ID	Title	Author(s)
1	1471	Local Scour Downstream of Type-A Trapezoidal Piano Key Weir	Chonoor Abdi Chooplou, Aram Ghafouri, Masoud Ghodsian, Mohammad Vaghefi
2	1483	Hydraulic Performance of Piano Key Weir with Linear and Curvilinear Outlet Keys	Prakash Kumar Sinha, Binit Kumar, Zulfequar Ahmad
3	1489	Experimental study on three-dimensional velocities near Piano Key weirs	Binit Kumar, Zulfequar Ahmad
4	1455	A Numerical Investigation on Residual Energy of Labyrinth Weirs	Phillip Langohr, Brian M. Crookston, Daniel B. Bung
5	1458	Introduction and investigation of a large-scale piano key weir fabricated via rapid prototyping	M. Oertel, B. Belay, X. Shen, H. Willems

Technical Session: TS-6	Date : 27/10/2022	Time: 1130 to 1300 Hrs
Theme: Sediment Transport	Location: L2-104, , A P J Abdul Kalam Block	

S No.	Paper ID	Title	Author(s)
1	1474	River aggradation-degradation under sand mining: experimental and numerical studies	Nejib Hassen Abdullahi, Zulfequar Ahmad
2	1452	Effect of Diameter and Inlet-depth on Hydro-suction performance of a Suction pipe	Akash Jaiswal, Zulfequar Ahmad, Surendra Kumar Mishra
3	1498	Bed load computation for gravel particles in gravel-silt mixture	Umesh K. Singh, Sanjeet Kumar, Z. Ahmad
4	1457	2-D Simulation of flow structures over dunes for flow characteristics estimation	Mohammad Sharifi, Mohammad Reza Majdzadeh Tabatabai, Seyed Hossein Ghoreishi Najafabadi
5	1500	Predicting riverbed deformation during floods for a sustainable design of headworks structures	Ishwar Joshi, Umesh Singh, Meg Bahadur Bishwakarma, Y. Kitamura, Leif Lia

Technical Session: TS-7		Date : 27/10/2022	Time: 1400 to 1530 Hrs
Theme: Physical and Numerical Modelling		Location: L2-103, A P J Abdul Kalam Block	
S No.	**Paper ID**	**Title**	**Author(s)**
1	1482	CFD-Based Modeling for Computing Discharge Coefficient of an Ogee Spillway	Hariom Gautam, Zulfequar Ahmad, Pramod Kumar Sharma
2	1454	Effect of sinuosity variation on flow characteristics of sand mined sinuous channel using numerical modeling	O.P. Maurya, K.K. Nandi, S. Dutta
3	1451	Estimation of Discharging Capacity and assessing the performance of Overflow Spillway with Full and Partial Operation of Gates using Numerical Modelling	Prajakta P. Gadge, Kunal Kapur
4	1426	Numerical simulation of air entrainment in stepped spillways	F. Zabaleta, S. Márquez Damián, F.A. Bombardelli
5	1492	SPH simulation of non-aerated flow over smooth invert, converging spillways	J.D. Nóbrega, J. Matos, H.E. Schulz & R.B. Canelas

Technical Session: TS-8		Date : 27/10/2022	Time: 1400 to 1530 Hrs
Theme: Scour around Hydraulic Structures		Location: L2-104, A P J Abdul Kalam Block	
S No.	**Paper ID**	**Title**	**Author(s)**
1	1461	Jet scour in cohesive and non-cohesive soils	Jessica Di Nardi, Michele Palermo, Stefano Pagliara, Fabian Bombardelli
2	1481	Scour downstream of a broad crested drowned weir	Rajat Chauhan, Rakesh Kumar Chaudhary, Zulfequar Ahmad
3	1494	Preliminary analysis of the effect of worked wood piles in straight channels	D. Roy, S. Pagliara
4	1499	Calculating local pier scour in coarse-bed streams	D.C. Froehlich
5	1439	Plunge pool rock scour assessment by fluid-solid coupling	E. F. R. Bollaert, M. Grünzner

Technical Session: TS-9		Date : 27/10/2022	Time: 1400 to 1530 Hrs
Theme: Training and Protection Works		Location: L2-005, A P J Abdul Kalam Block	
S No.	**Paper ID**	**Title**	**Author(s)**
1	1441	Bank scour protection using spur dyke in a meandering channel under low flow velocity	Nishank Agrawal Ellora Padhi
2	1477	Numerical modeling for optimization of the aspect ratio of submerged vanes for the purpose of sediment deflection in rivers	V. Chauhan R. Chavan G.D. Singhal
3	1475	Experimental evaluation of drag force on different shapes of pontoons at different water stream velocities at specified loads and submergence	R. B. Deogade, H. R. Khandagale, Shri. M. Someshwara
4	1476	Performance testing of acoustic doppler current profiler used for stream flow measurement	R. B. Deogade, H. R. Khandagale, Shri. M. Someshwara
5	1447	Porous Concrete Slabs Underlain by Polypropylene Fabric for Restoration of Beaches	N.N. Pillai, M.L. Kansal, R. Ramkrishnan

Technical Session: TS-10	Date : 27/10/2022	Time: 1600 to 1730 Hrs
Theme: Physical and Numerical Modelling	Location: L2-103, A P J Abdul Kalam Block	

S No.	Paper ID	Title	Author(s)
1	1501	Predicting Density-driven Exchange Flows through the West Crack Breach of the Great Salt Lake Causeway with CFD and ANN	B.M. Crookston, M. Rasmussen, E. Larsen, H. Kartchner, S. Dutta
2	1433	Hydrodynamics of Permeable Horseshoe Obstacle in Fluvial Environment: a Physical Modelling	Hubert Chanson, Hoit Kit Lee, William Johnson
3	1434	Near-full-scale physical modelling and open-channel flow velocity in a fish-friendly culvert with full-height sidewall baffles	Jiayue Hu, Youkai Li, Hubert Chanson
4	1484	Physical and numerical model studies of Hirakud dam additional spillway –a case study	Vaishali P. Gadhe, S.R. Patnaik
5	1449	Flow over orifice spillway: Physical and numerical study for spillway profile design of hydroelectric project	Sushma Vyas, Y. N. Srivastava

Technical Session: TS-11	Date : 27/10/2022	Time: 1600 to 1730 Hrs
Theme: Scour around Hydraulic Structures	Location: L2-104, A P J Abdul Kalam Block	

S No.	Paper ID	Title	Author(s)
1	1470	Laws of turbulence and the estimation of turbulent kinetic energy budget for flow through a degraded channel-bed	Mosedul Sarkar, Sankar Sarkar
2	1464	Numerical modelling of scour and flow field for two different arrangements of piers using SSIIM model	Sushant Kumar Biswal, Animesh Das
3	1493	Scour geometry and dune formation characteristics around customised structure in channels	Rajib Das, Subhasish Das, Michele Palermo, Deep Roy, Simone Pagliara
4	1491	Estimation of maximum scour depth and scour pattern around submerged spur-dyke	Manish Pandey, Manali Pal
5	1467	An experimental study on scour at zero degree confluent channels	Mohd Faisal Ansari, Mohammad Aamir, Zulfequar Ahmad

Technical Session: TS-12	Date : 27/10/2022	Time: 1600 to 1730 Hrs
Theme: Miscellaneous	Location: L2-005, A P J Abdul Kalam Block	

S No.	Paper ID	Title	Author(s)
1	1444	Experimental optimization of a gravel trap in a gallery	S. Erpicum, N. Claude, B. Dewals, P. Archambeau, M. Pirotton
2	1438	Investigation on near bed flow features over a water-worked gravel bed	Ellora Padhi, Nishank Agrawal
3	1436	Reynolds Stress Modelling of Supercritical Flow in a Narrow Channel	S. Kadia, N. Rüther, E. Pummer
4	1453	Importance of risk and hazard assessment of river projects in India	Jitesh N. Vyas, Supriya Nath and R. B. Deogade
5	1430	Designing Smooth Mixed-Geometry Canal Transition	D.C. Froehlich

THEMES OF THE SYMPOSIUM

Storage and Diversion Structures

Dams and Weirs
Spillways
Intakes and outlets
Fish passes
Navigational locks
Energy Dissipators
Stilling basins
Block ramps
Stepped spillways
Plunge pools

Flow Conveyance Structures

Canals, tunnels, pipes
Penstocks & surge tanks
Flood mitigation channels
Gates and Valves

Physical and Numerical Modelling

Instrumentation
Scale effects
Modelling of roughness & sediment
Multiphase modelling
Hybrid modelling
Fluid structure interaction
Case studies and prototype measurements

Scour around Hydraulics Structures

Scour and sediment transport
Aggradation and degradation
River training and protection works

Coastal Engineering

Ports and harbours
Coastal and offshore structures
Caisson and rubble mound breakwaters

Best Practices in Risk Management

Dam safety and rehabilitation
Risk and hazard assessment
Sustainable design
Adaptation to climate change

Miscellaneous Structures

Non-linear weirs
Pump sump
Trench/Tyrolian weirs
Permeable weirs

Reservoir Sedimentation


Tehri Dam, India


Maithon Dam, India

TECHNICAL TOUR

The technical tour is a key component of the symposium and tentatively will include visit to Tehri Dam and Upper Ganga Canal.

A. TEHRI DAM

Date and Start time: 24th October, 2022; 8:00 AM

Assembly Point : Foyer of Dept. of Civil Engineering


Tehri Dam, India

Tehri Dam is one of the highest dams of its type in the world on the Bhagirathi River near Tehri in Uttarakhand, India. Its length is 575 m, crest width 20 m, and base width 1,128 m. The dam creates a reservoir of 4.0 km^3 with a surface area of 52 km^2. It comprises of a 260.5 m high Earth & Rockfill dam, a Spillway System having one Chute Spillway and four Shaft Spillways designed for PMF of 15540 m^3/s and a drop of 220 m and an underground Power House housing four Turbine/Generator sets of 250 MW each, designed to operate with a head variation of 90 m. The project was commissioned in 2006-07 and all four machines of Tehri Power Station are under commercial operation. Besides providing much needed power to the Northern Grid, the command area is availing irrigation benefits from the Project and drinking water is being supplied to various state of India.

B. Upper Ganga Canal

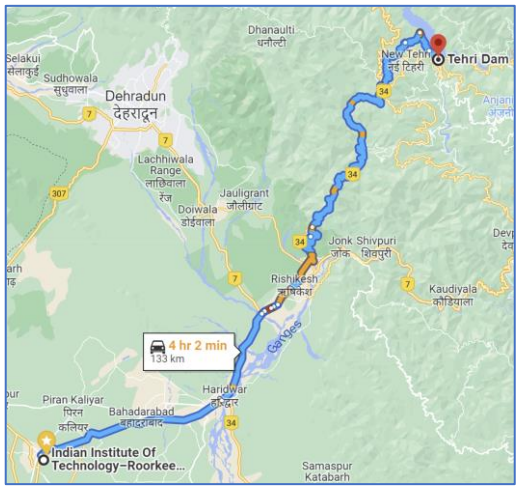
Date and Start time: 24th October, 2022; 8:00 AM

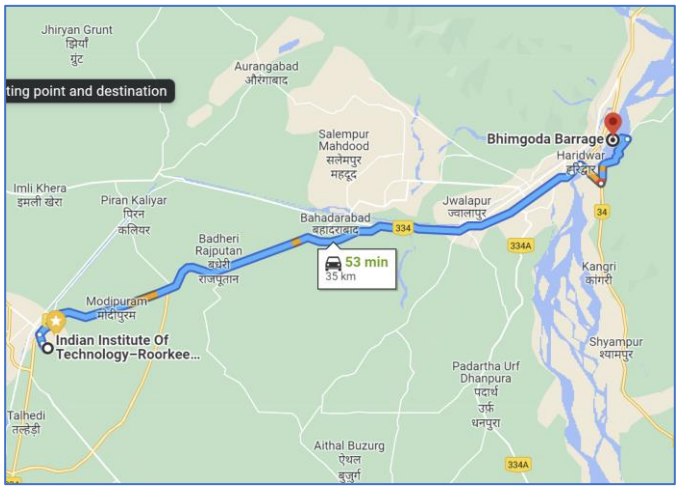
Assembly Point : Foyer of Dept. of Civil Engineering

After the disastrous Agra famine of year 1837–38, in which nearly 800,000 people died, Colonel Proby Cautley, who has been affectionately remembered as a British Engineer with an Indian heart, conceived and constructed a canal irrigation system known as Upper Ganga Canal during the period 1840-1854. Its command area is 24000 km^2 and augmented flow is 297 m^3/s. The system consists of a main canal of

272 miles and about 4000 miles long distribution channels. The Upper Ganga Canal was the largest and costliest man-made waterway in the world in its opening year 1854. It is aligned along contour line in its upper reaches due to which it crosses other rivers and accordingly cross-drainage works are provided. Acciavatti writes in his book that engineers came from around the world to see such cross drainage works which outdid any canals and aqueducts that had been built before. Four major cross-drainage works are Ranipur Syphon, Pathri Super Passage, Dhanuri Level Crossing and Solani Aqueduct which was ranked as one of the most remarkable massive brick masonry structure in the whole world. It was to help the construction of this canal, reason why the first engineering college in India, the Thomason College of Civil Engineering was set up at Roorkee which was later converted into IIT Roorkee in the year 2001.

Solani Aqueduct, India





WORKSHOP

A workshop is organised on 25th October 2022 in the Conference room of the Department of Civil Engineering on the sustainable design and construction of nonlinear weirs. Prof. Sebastien Erpicum and Prof. Brian M. Crookston shall conduct this symposium.

Sebastien Erpicum
Liege University, Belgium

Brian M. Crookston
Utah State University, USA

Dr. Erpicum has been one of the key pioneers in researching, understanding, and publishing information related to the development and implementation of the innovative Piano Key Weir concept and its application to several dam rehabilitation projects in France and abroad.

Dr. Brian consulting and research interests are focused on water conveyance, including the design of hydraulic structures, nonlinear spillways, flow acoustics, energy dissipators, scour, fish passage, and physical and numerical modeling.

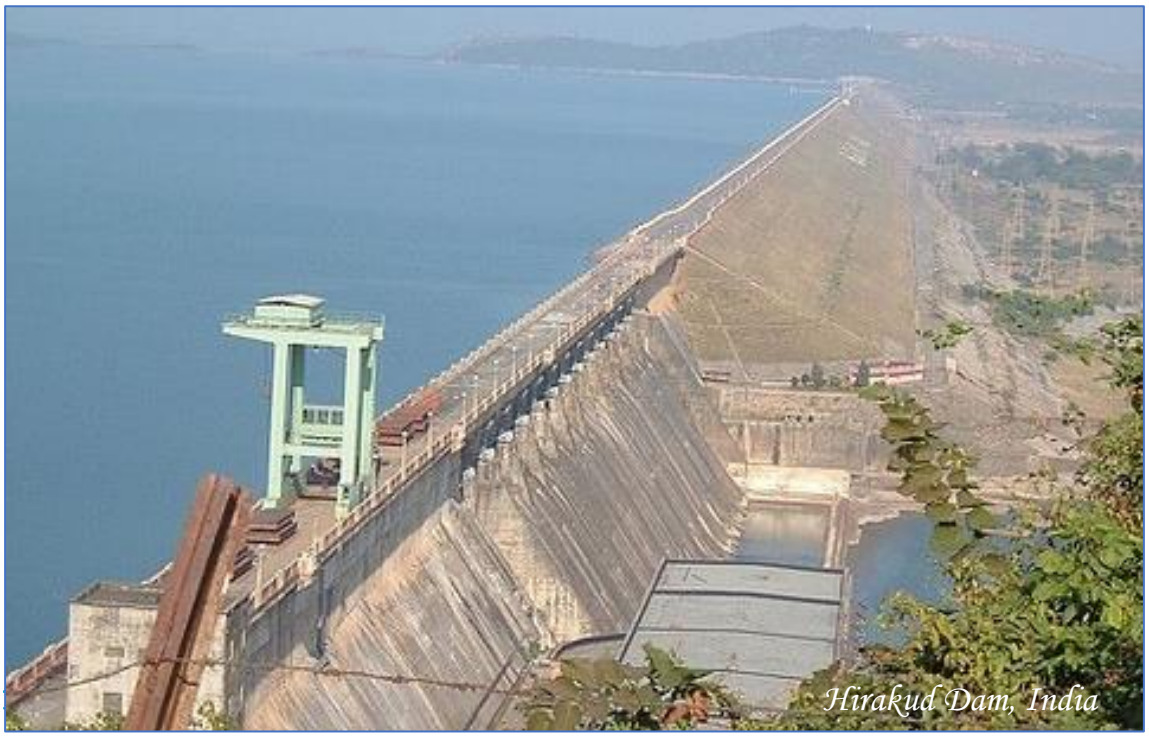
ABSTRACT

Inadequate conveyance capacity, significant hydrologic loadings on embankments and structures, and operation and maintenance are common challenges and safety issues that often instigate the need for risk-reduction measures and/or rehabilitation. Nonlinear weirs such as labyrinth and piano key weirs are a particularly efficient approach to address these concerns. These weirs provide a crest length greater than the width of the channel and are commonly used in a variety of applications, including run-of-river structures, dam rehabilitation and new dam projects. In some cases, these structures may be complimentary to or even an alternative to gated

spillways, which require operation and maintenance and can cause downstream flooding if mis operation or failure occur. Because of their hydraulic performance and site-adaptive geometries, these types of weirs are of increasing interest to those involved in water infrastructure, including practitioners, researchers, regulators, and owners. However, these types of weirs have complex geometries and hydraulic behaviors that pose a challenge when developing an efficient design.

The workshop will cover all relevant stages of a labyrinth and a piano key weir project, from the first theoretical design to detailed studies and practical considerations related to the construction. The workshop will be divided in two parts, each related to a specific type of weir. During both parts, several real examples in France and the USA will illustrate the technical presentations and time will be devoted to Q/A and discussion.

Idukki Dam, India



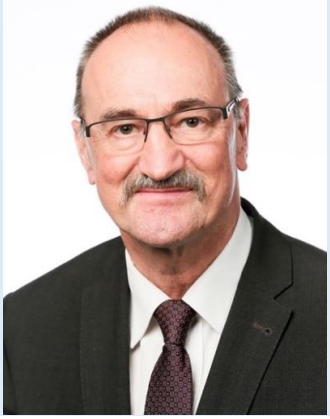
Hirakud Dam, India

KEYNOTE SPEAKERS

Six eminent researchers and practitioners shall deliver keynote lectures on various topics of the Hydraulic Structures as per detail below:

Anton J. Schleiss

Professor emeritus EPFL and independent expert consultant, Switzerland



Title: The Challenge of Rock Scour Assessment at High Head Spillways

ABSTRACT

In today's spillway design of dams there is a tendency of increasing the unit discharge of high-velocity jets leaving the appurtenant water release structures. For gated chute flip bucket (ski jump) spillways, unit discharges between 200 and 300 $m^3/s/m$ are not rare anymore, since the cavitation risk is mitigated by chute bottom aerators. Crest spillways for arch dams are currently designed for unit discharges from 70 $m^3/s.m$ to 120 m^3 /s.m by installing crest gates. With the latest high-pressure gate technology, mid- and low-level orifice spillways evacuate unit discharges up to 400 $m^3/s.m$. Scour of rock is of concern when assessing the effect of jets released from spillways plunging onto rock foundations downstream of dams or designing plunge pools. Rock scour is a complex problem studied extensively experimentally but often in a simplified way by using granular material. Nevertheless, reliable scour prediction in rock must consider the relevant physical processes when fluctuating turbulent pressures of highly aerated flow in the plunge pool are propagating into fractured rock masses resulting in a strong fluid-air-rock interaction. In modern scour evaluation methods, the potential and extent of rock scour is determined by employing the principles of fracture mechanics and Newton's second law. Continuing research into various aspects of turbulent flow and its interaction with rock formations refine this procedure. The main difficulties encountered when estimating scour depths are discussed, including the choice of the appropriate theory for rock scour assessment, interpretation of hydraulic model tests and prototype observations, the scour rate, and the prevailing discharge. The selection of the flood return period is addressed, for which the scour formation and the control measures have to be evaluated, as also options of measures for scour control. Finally, the challenges for dam designers are

presented when answering relevant questions reading dam safety and powerhouse operation.

ABOUT THE SPEAKER

Prof. Dr Anton J. Schleiss graduated in Civil Engineering and obtained a PhD on the topic of pressure tunnel design from ETH in Zurich, Switzerland. After having worked for 11 years with Elektrowatt (now AFRY) in Zurich, he was nominated 1997 full professor and director of the Laboratory of Hydraulic Constructions of EPFL in Lausanne. He supervised more than 50 PhD and Postdoc research projects. In 2006, he obtained the ASCE Karl Emil Hilgard Hydraulic Price as well as the J. C. Stevens Award. He was listed in 2011 among the 20 international personalities that "have made the biggest difference to the sector Water Power & Dam Construction over the last 10 years". In 2015, he obtained the ASCE-EWRI Hydraulic Structures Medal. The French Hydro Society (SHF) awarded him with the Grand Prix SHF 2018 and IAHR in 2021 with the honorary membership. After having served as vice-president since 2012, he was president of the International Commission on Large Dams (ICOLD) from 2015 to 2018. With more than 40 years of experience he is regularly involved as a consultant and expert in large water infrastructures projects including hydropower and dams all over the world.

Mike Phillips

Senior Hydraulic Structures Engineer.
Risk Management Center
US Army Corps of Engineers - Institute for Water Resources

Title: Modifying the Mosul Dam Bottom Outlet and Flip Bucket

ABSTRACT

Mosul Dam is located in Nineva, Iraq on the Tigris River, and is owned by the Government of Iraq and operated and maintained by the Ministry of Water Resources (MoWR). The dam is a 110 m high earth and rockfill embankment with primary purposes of water supply, hydropower, and flood control. The Mosul Dam Bottom Outlets were designed to divert the river during construction, to manage the reservoir when the lake level is sufficiently low that hydropower cannot be generated, or to lower the reservoir during emergencies. The bottom outlet consists of a submerged

intake tower which bifurcates the flow into two identical conduits, each with 12 m diameter reinforced concrete culvert sections, an access and emergency regulation tower, followed by 10 m diameter steel lined tunnel sections, and regulated by 7 m high by 5 m wide top seal tainter gates. Each bottom outlet has a maximum discharge capacity of 1,225 m^3/s with 55 m of reservoir head. The energy dissipation structure for the bottom outlets consists of a flip bucket located downstream of the tainter gates and a partially concrete lined plunge pool. This case study presents a brief synopsis of the Mosul Dam bottom outlet, a brief history of the project, the issues regarding operation of the bottom outlet, erosion issues in the plunge pool, and the hydraulic issues related to the design, construction and commissioning of the new dentated flip bucket to reduce the potential for erosion in the plunge pool.

ABOUT THE SPEAKER

Mr. Phillips is a Senior Hydraulic Structures Engineer with 22 years of experience in civil engineering, with expertise in hydraulic structures engineering (hydraulics and structural), dam safety, water resources, and risk management. He serves as a National and International subject matter expert for USACE in hydraulic structures engineering across a range of dam and water resources related projects (domestic and international civil projects, interagency working groups, and military). Mr. Phillips has served as the hydraulic structures design engineer, project manager, advisor, or risk assessor on spillways, outlet works, and appurtenant structures for more than 100 water storage dams, flood protection levees, flood detention dams, and mine tailings projects in the USA, Australia, New Zealand, Peru, Colombia, Iraq, Turkey, Saudi Arabia, Uzbekistan, Ecuador, and South Korea.

Sebastien Erpicum

Liege University, Belgium

Title: Composite Modeling for Hydraulic Structures Design

ABSTRACT

Modeling plays a central role in hydraulic structures design since it is the only method able to analyse such large structures behavior in normal and extreme conditions. The

combined application of physical and numerical modelling, so called composite or hybrid modeling, is widely recognized as the most effective strategy for the in-depth analysis of complex flow and transport processes, both in basic research and for the design of real-world projects. It aims at capitalizing on the benefits of a synergetic implementation of physical and numerical modelling as two highly complementary components. In this lecture, several strategies that can be applied to enhance hydraulic structures design by means of a composite modeling approach are detailed, and the benefit arising from composite modeling is emphasized. It is shown that there is no question of choice between physical and numerical modeling in hydraulic structures design. None of the two approaches enables a perfect prediction of flow conditions on the real structure. However, combined application through composite modeling is a way to decrease the uncertainties linked to each approach individually.

ABOUT THE SPEAKER

Dr Sébastien Erpicum is Associate Professor at Liege University, Belgium, where he is the head of research and modelling activities at the Engineering Hydraulics Laboratory. Starting from a strong international background in the use of physical scale hydraulic modeling as a problem-solving tool, he expanded the laboratory capability through the development of composite modeling techniques (coupled physical and numerical modeling). In association with three colleagues (Dr. P. Archambeau, Prof. B. Dewals and Prof. M. Pirotton), he founded the Hydraulics in Environmental and Civil Engineering (HECE) research group, which develops the modeling system WOLF and conducts research ranging from hydrology to hydraulic structures engineering, and flood damage evaluation.

Dr Erpicum is past chair of the IAHR Hydraulic Structures Committee and Belgian representative at the Hydraulics for dams Technical Committee of ICOLD. He received the ASCE-EWRI Hydraulic Structures Medal in 2020 and the 22nd Arthur Ippen award from IAHR in 2021.

His recent research topics, closely related to hydraulics and hydraulic structures, concern the design and operation of spillways, in particular nonlinear weirs, small hydropower and its interaction with fish migration, and also energy storage.

Erik Mosselman

An expert in river engineering and fluvial morphodynamics, Deltares

Title: Experiences from training the Brahmaputra-Jamuna River

ABSTRACT

The Brahmaputra-Jamuna in India and Bangladesh is a unique river. It is about 10 km wide and conveys discharges that average 20,000 m3/s and can peak at 100,000 m3/s. Its morphology is so dynamic that channels and banklines can shift hundreds of metres in a single year. All this poses great challenges to river training for safety against flooding, freshwater supply, infrastructure crossing and navigation. River training projects and associated studies over the last 30 years have produces a rich array of experiences. These experiences include technological advances regarding river training structures, data collection and modelling, but they are not limited to that. Key experiences also regard insights in long-term sustainability and principles for the development of master plans.

ABOUT THE SPEAKER

Erik Mosselman is a river scientist and hydraulic engineer with special expertise in fluvial morphodynamics, river training, ecological river restoration and flood risk management. He holds permanent positions at Deltares (0.7 fte) and Delft University of Technology (0.3 fte). He worked on numerous rivers in Europe, Asia, Africa and the Americas, with long-standing involvement in the Room-for-the-River programme for the Rhine branches in the Netherlands, development of a sustainable inland waterway between the Port of Rotterdam and Germany, and stabilization of the Brahmaputra-Jamuna River in Bangladesh.

Stefano Pagliara

Professor of Hydraulic
Construction, University of Pisa, Italy

Title: Hydraulic Behaviour of In-Stream Low Head Structures for River Restoration.

ABSTRACT

Small hydraulic structures such In-stream grade-control structures are used to protect riverbanks, and to improve and rebuild aquatic habitats. In the last decades many of these structures have been analyzed. Among the others, block ramp, cross vane, j-hook, chevron type and wood logs.

All these structures have a great value for river restoration. The paper will show some experimental studies that highlight the behaviour of these structures.

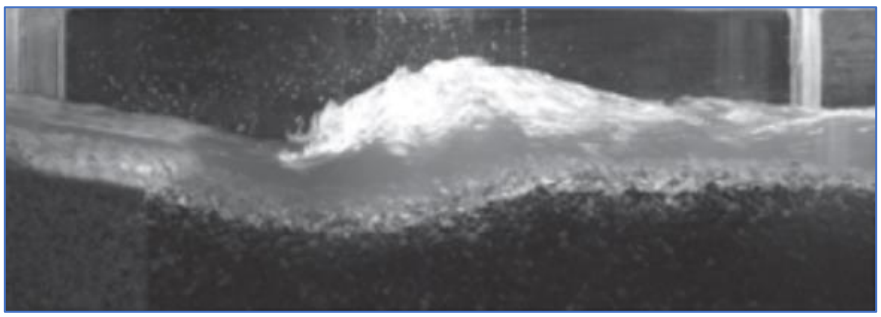


Fig.1 Basin of a block ramp

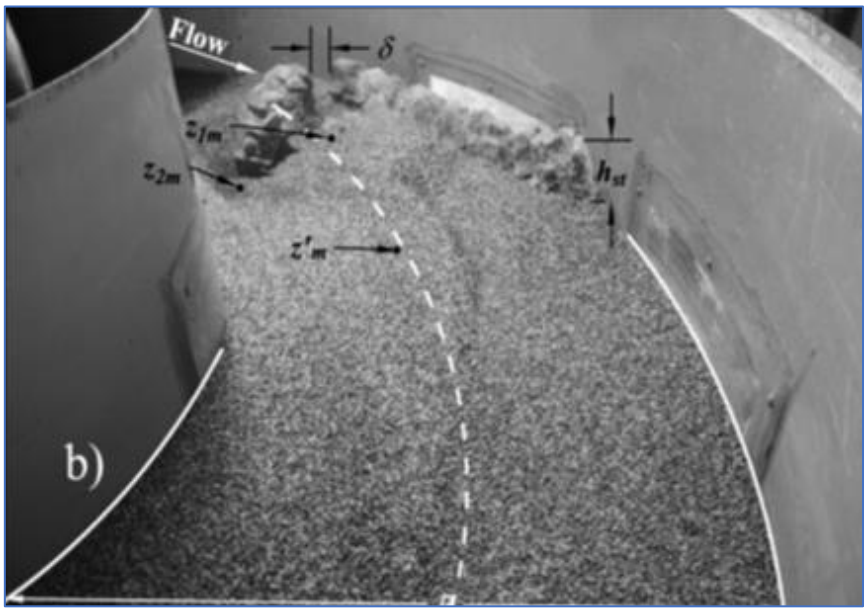


Fig. 2 J-hook in ba bend channel

Fig. 3 Wood log frame

The lecture will show the study of the scour phenomena and morphologies downstream of these structures. Experiments have been conducted in clear-water conditions. The results showed the important effect on the position in the channel (straight or curved) and the effect of the structures angles and slopes.

One important parameter is the protection of the side banks from the scour process. Useful empirical relationships have been proposed in order to evaluate the main features of the scour geometry for all the tested situation.

A comparison between different structures type has been done in order to show pro and contras of each geometry.

Among the analyzed structures some are made of wood. This is an important advance in the construction of these small structures due to the low environmental impact and of their ecological use.

ABOUT THE SPEAKER

Stefano Pagliara is full professor of Hydraulic structures at the University of Pisa, Italy. He has been Fulbright researcher at the University of Illinois at Urbana/Champaign and researcher for the Japanese government at the Public Work Research Institute at Tsukuba - Ibaraki – Japan. He has been Visiting Professor at ETH Zurich (CH). He is author of 300 papers mainly in international journals, board member of IAHR and chairman of the Hydraulic structure committee of IAHR. His research interests are on hydraulic structures, jet scour, river restoration, urban drainage and flood risk and mitigation.

Fabian A. Bombardelli

Department of Civil and Environmental Engineering, University of California, Davis

Title: Fluid mechanics past hydraulic structures: A shift in paradigm

ABSTRACT

Traditionally, flows past hydraulic structures have been addressed from the design point of view. More specifically, the concern has been chiefly to provide heights of walls, widths of channels, etc., without much regard to the internal features of the flow. Design formulas have been experimentally obtained for the inception point of air entrainment, for instance, and they have helped develop the hydraulic profession. Very recently, on the other hand, high-fidelity numerical simulations have opened a new avenue, which consists in addressing the turbulent coherent structures leading to the interpretation of flows past hydraulic structures. These simulations are shedding light on the flow features which explain air entrainment, describe eddies upstream and downstream gates, for example, and offer new ways of "seeing" the hydraulic structures. In this presentation, diverse solutions will be discussed, indicating how they compare with past and new observations, including field data, and how they can be used to finally improve the design of those structures.

ABOUT THE SPEAKER

Dr. Fabián A. Bombardelli holds the Gerald T. and Lillian P. Orlob Endowed Professorship in Water Resources, at the Department of Civil and Environmental Engineering of the University of California, Davis (UC Davis). Dr. Bombardelli is a leader in the development of new theoretical and numerical models for multi-phase flows, as well as in their observation in the laboratory and the field. He currently serves as the Editor in Chief of the Journal of Hydraulic Engineering, of the American Society of Civil Engineers (ASCE), and of RIBAGUA, the International Journal of Water of Iberoamerica, IAHR. Dr. Bombardelli received a degree in Hydraulic Engineering from the National University of La Plata, Argentina; a Master degree in "Numerical Simulation and Control" by the University of Buenos Aires, also in Argentina; and a PhD by the University of Illinois, Urbana-Champaign, United States, under the supervision of Prof. Marcelo Garcia. Prior to his move to the States, he was a Researcher in Numerical Models at the National Water Institute for seven years. Since 2004, Dr. Bombardelli is a Professor (now Full with tenure) at the UC Davis.

Dr. Bombardelli is widely known for his research on bubble plumes, sediment transport in open channels, the Basset force, flow in stepped spillways, and for the application of the phenomenological theory of turbulence to hydraulics; in addition, he has developed applied research on water bodies in California. He has published in major research journals of physics, hydraulic engineering and water resources. He has more than 70 publications in these journals and more than140 articles in total. Dr. Bombardelli is a member of the Editorial Board of the Journal Environmental Fluid Mechanics since 2011; Associate Editor of the Journal of Hydro-environment and Research, since 2018; and member of the Review Committee of the Int. Journal of Sediment Research. He has received numerous recognitions such as the Best Reviewer Award of the IAHR (2011), Outstanding Reviewer of the ASCE (2011), the Awards as Outstanding Advisor in Civil Engineering (ASCE, 2015), and Outstanding Advisor in Civil Engineering of the State of California (2015), the Young Alumni Award of the University of Illinois in 2015, Featured Article in Physics of Fluids (2018), EWRI Fellow in 2021, and various awards as a student in Illinois. 12 students have graduated with doctorates and 27 as Master under the supervision of Prof. Bombardelli. He has also worked as a consultant for the government of Argentina and for the United Nations in Peru, in 2011 and 2013, proposing systems of cascades for the very polluted Matanza-Riachuelo, Buenos Aires. He has delivered seminars and keynote lectures in many universities and conferences worldwide.


Barki River, India

ABSTRACT OF THE PAPERS

The abstract of the 60 selected papers is given below and the full-length papers shall be avail published in the Proceedings, made freely available online, and distributed electronically to attendees. The proceedings will be indexed in Scopus and made available online at Utah State University Digital Commons.

Durgapur Barrage, India

Panchet dam, India

Paper ID: 1432

Surface Velocities and Free-surface Aeration in a Converging Smooth Chute during a Major Flood Event

H. Chanson[1] & C.J. Apelt[1]
[1]*The University of Queensland, School of Civil Engineering, Brisbane, Australia*
E-mail: h.chanson@uq.edu.au

Abstract: *In a free-surface spillway, the upstream flow is non-aerated and the flow becomes a strong air-water mix downstream of the onset location of air entrapment. Field observations were conducted over a large dam's spillway during a major flood event. The wastewaterway system was a smooth converging chute with a longitudinal slope of 11.3°: the Chinchilla Minimum Energy Loss weir on the Condamine River (Australia). Detailed quantitative measurements were undertaken in the high-speed chute flows with strong turbulence and high Reynolds numbers at the early part and later part of a major flood event. During this exceptional flood, the spillway passed nearly three times the design discharge capacity at the peak for the event. Down the smooth chute, the observations indicated that the onset of free-surface aeration was a complicated transient three-dimensional process. A robust optical flow (OF) technique was applied and delivered physically-meaningful surface velocities in the air-water flow region. The streamwise surface velocities were reasonably close to backwater calculations. The data presented large streamwise surface velocity fluctuations in the aerated flow region, with* Tu_s *~ 150-200%, on the centreline consistent with self-aerated flow measurements using dual-tip phase detection probe in laboratory. Overall, the study demonstrated the application of optical techniques to prototype smooth spillway flows, as well as some intrinsic difficulties with field investigations.*

Keywords: *Prototype velocity measurements; Smooth converging chute; Self-aeration; Optical techniques; Minimum Energy Loss weir.*

Paper ID: 1459

Feasibility Assessment of the Application of Delft3D Model to Simulate Flow over a Spillway in a Dam: Study Case of Hampaturi Dam in Bolivia

F.D. Oliveira[1] & M.W. Heredia[1]
[1]*Hydraulics Laboratory, Universidad Mayor de San Simón, Cochabamba, Bolivia*
E-mail: marceloheredia.g@fcyt.umss.edu.bo

Abstract: *Numerical modelling is an attractive alternative over physical modelling. However, sometimes both kind of techniques are coupled. This is the case of Hampaturi Dam in La Paz, Bolivia. A physical model was built for the optimization of the spillway. Hence, a numerical model was implemented with Delft3D computational model. Delft3D is a model based on two-dimensional flow equations (hydrostatic equations), and full three-dimensional equations (hydrostatic equations). Both modes were tested for the flow over the spillway, which includes a steep discharge channel that poses an additional difficulty. The goal of this research is to test the model (in both modes) on these flow conditions in order to determine the feasibility in applying Delft3D on this kind of structure. The model was calibrated and validated for both modes, hydrostatic and non-hydrostatic, by the variation of Manning's roughness coefficient and eddy viscosity, reproducing the observed flow velocities and water depths. Following, a sensitivity analysis was carried out to determine the influence of different parameters in both modes, like: number of vertical layers, bed roughness and eddy viscosity. This analysis was executed by using an independent mesh solution. Results show a significant influence of the tested parameters. Then, the influence of the channel steepness on the performance of the model was tested. Results show that non-hydrostatic mode produces smooth flow profiles. Finally, it was*

observed that both modes can operate up to inclination angles of 60° for the discharge channel, setting an operation limit for Delft3D in this kind of structure.

Keywords: *Spillway dam, Delft3D model, three-dimensional model, hydrostatic, non-hydrostatic, sensitivity analysis.*

Paper ID: 1442

Hydraulic Behavior of Spill channel of Polavaram dam spillway without left embankment

Vankayalapati S. Ramarao[1] Yogendra Nath Srivastava[2]
[1] *Scientist -C, Central Water and Power Research Station, Khadakwasla, Pune-411 024. India*
[2] *Additional Director, Central Water and Power Research Station, Khadakwasla, Pune-411 024, India*
Email: sramaraov@gmail.com

Abstract: *Spill channel is a flood disposing channel excavated on the downstream of spillway which joins the main river course. The purpose of spill channel is to divert the spillway released flood safely to the downstream river course. The alignment of the spill channel may be straight of curved or partially straight and partially curved. It may be in steep gradient or in flat gradient and subjected to supercritical flows or subcritical flows. While disposing of the flood, the flow conditions in the spill channel would be straight without cross waves. Water surface profiles in the convex and concave banks of the spill channel must be confined within the top elevation of the embankments of channel to avoid spilling over the embankments. Thus, the hydraulic design of the spill channel involves keeping the velocities in the channel to allowable limits to prevent erosion of embankments and also confining the water levels to within the top elevation of the embankments. Suitable foundations are necessary to keep the embankments of spill channel stable. If a suitable foundation is not available for embankments, the embankment sometimes may have to be removed, if it would not cause problem in disposing of the desired quantum of flood. Not much literature is available relating to these kinds of studies carried out on large spill channels carrying more than 1.41 lakh m^3/s discharge and without one embankment. The general procedure for selection of optimum channel parameters is of trial and error by considering number of alternatives and the hydraulic model studies play a key role in optimising the design of spill channel. Hydraulic model studies of Polavaram Irrigation Project, Andhra Pradesh were carried out in CWPRS, Pune on 1:140 scale 3-D comprehensive model to assess the subcritical flow conditions in spill channel with and without the provision of left embankment for various magnitudes of flood ranging upto 141583 m^3/s. Studies enabled optimisation of layout of spill channel to pass the design flood with uniform flow conditions in the spill channel. In this paper, the various aspects of optimization of the hydraulic design of spill channel in disposing of the flood safely to downstream, based on the results of hydraulic model studies are discussed.*

Keywords*: confluence, flood, guide bund, hydraulic design, spill channel, spillway.*

Paper ID: 1479

Recent challenges in design of spillway – An Indian scenario

R.R. Bhate[1]
[1]*Scientist 'C', Central Water and Power Research Station, Pune, India*
E-mail: ravindra.bhate@gmail.com

Abstract*: There are about 5264 completed large dams in India. These dams are invariably equipped with some or other kind of spillway. As this dam building activity has been going on for many years, all easy sites have already been tapped and are left with sites where designers have to face challenges from*

nature. Recently, while harnessing the hydropower potential of Himalayan rivers, designers are facing issues like high sedimentation concentration flows, narrow valleys with a high magnitude of the flood, leading to the provision of an additional spillway or multi-tier spillways. Melting of glaciers resulting in increased rates of Glacial Lake Outburst Flow (GLOF) is challenging tasks to design a spillway for debris flow. On the other hand, for existing dams in India, there is a large upward revision in the design flood due to improvements in flood assessment techniques and the availability of huge data for flood assessment. This demands an increase in discharging capacity of the spillways by increasing crest length or providing breaching sections. India has got a large seashore and to utilize the freshwater from the rivers joining the sea, multipurpose dams across the sea in the vicinity of the Gulf can be considered. Special design considerations are to be employed for spillway and energy dissipation arrangements as their operational performance is influenced by cyclic tidal variations and storm surges. This paper makes an attempt to discuss the relevant hydraulic aspects for the design of spillways and energy dissipators for efficient and safe operation.

Keywords: *Orifice Spillway, Sediments, Flood, GLOF, Coastal reservoir, Hydraulic model*

Paper ID: 1435

Non-intrusive measurements of air-water flow properties in supercritical flows down grass-lined spillways

H. Cui[1], S. Felder[2] & M. Kramer[1]
[1]*UNSW Canberra, School of Engineering and Information Technology (SEIT), Canberra, ACT 2610, Australia*
[2]*Water Research Laboratory, School of Civil and Environmental Engineering, UNSW Sydney, Manly Vale 2093, NSW, Australia*
E-mail: m.kramer@adfa.edu.au

Abstract: *Grass-lined spillways constitute an environmentally friendly alternative to conventional concrete structures with flat slope and therefore may contribute to water-sensitive urban design. Research into supercritical self-aerated flows down grass-lined spillways has commenced recently, but important knowledge gaps related to air entrainment, flow resistance and bubble-canopy interaction are yet to be filled. This study presents first results of an experimental investigation into flows over a rough grass-lined channel bed with embankment slope. The deployed measurement instruments included acoustic displacement meters (ADMs) and a conventional intrusive phase-detection probe. A transformation of measured free-surface elevations into multiple time series of instantaneous air concentrations allowed to non-intrusively compute air concentration distributions, dimensionless interface frequency distributions, and chord times. The results demonstrated that the amount of entrapped air was significantly larger than the amount of entrained air. It was further shown that air concentration distributions followed a recently developed semi-analytical solution with a characteristic length that was closely linked with the mean air concentration. The ADM measurements were limited to entrapped air and future research should explore other non-intrusive instrumentation to also explore measurements of entrained air.*

Keywords: *Grass-lined spillway, air entrainment, non-intrusive sensing technology, acoustic displacement meter, entrapped air, interface frequency*

Paper ID: 1446

Model Studies for Desilting Basin for Teesta-VI H.E. Project, Sikkim – A case study

M.Z. Qamar[1], M.K. Verma[2], A.P. Meshram[3] & Dr. Neena Isaac[4]
[1]*Scientist 'C', Central Water and Power Research Station, Pune, India*
[2]*Scientist 'D', Central Water and Power Research Station, Pune, India*
[3]*Assistant Research Officer, Central Water and Power Research Station, Pune, India*

[4]*Scientist 'E', Central Water and Power Research Station, Pune, India*
E-mail: ziaul_qamar@rediffmail.com

Abstract: *Desilting basins play an important role in run-of-river hydro power projects on Himalayan Rivers. These rivers carry huge amount of sediment with them, due to steep slopes and fragile geology of the region. The suspended sediment enters through the power intake and ultimately the power house. This causes heavy damage to the turbines and other under water parts. Therefore, desilting basins are provided to eliminate suspended sediment from the water conductor system. The design of desilting basin is verified on a physical model for 90% removal of suspended sediment coarser than 0.2 or 0.3 mm and efficacy of flushing tunnel below in transporting the settled sediment. Central Water and Power Research Station, Pune has conducted physical model studies for desilting basin for various hydro power projects in India and neighbouring countries. One such study, Hydraulic model studies for desilting basin for 500 MW Teesta-VI Hydro Electric Project, Sikkim is presented in this paper. Various design parameters such as length of desilting basin, length and bed slope of inlet transition, outlet transition, size of silt flushing tunnel, size and spacing of openings connecting main basin with silt flushing tunnel etc. were tested during the model studies. The results in terms of settling efficiency for 0.3 mm size particles were obtained.*

Keyword: *Desilting basin, suspended sediment, settling efficiency, silt flushing tunnel.*

Paper ID: 1472

Hydrodynamic behavior of stilling basins with deflectors to reduce supersaturation of total dissolved gases: analysis of Yacyretá and Jorge Cepernic dams

C.A. Fattor[1], J.D. Bacchiega[1], F. Romero[1] & C. Fernández[1]
[1]*Hydraulic Structures Division, Hydraulics Laboratory, National Institute of Water, Ezeiza, Argentina*
E-mail: cfattor@ina.gob.ar

Abstract*: High air entrainment to the hydraulic jump is one of the inherent aspects of this macroturbulent flow, extending its presence from the free surface to the bottom. Depending on the variety of fishes in this natural habitat, important values of supersaturation of total dissolved gases (TDG) could be reached immediately downstream of the energy dissipation structure, putting under risk the life of fishes. This article deals with the influence of deflectors designed to reduce TDG on the hydrodynamic behavior of energy dissipators of two important dams. The first one is Yacyretá dam (Paraná River, Argentina – Paraguay) where horizontal deflectors were installed after an environmental incident with important mortality of fishes due to supersaturation of TDG. The second case corresponds to Jorge Cepernic dam (Santa Cruz River, Argentina), which is currently under construction. Hydrodynamic evaluation of the stilling basins under initial design conditions and those derived from the installation of the deflectors was carried out by means of two-dimensional physical models of each of the mentioned projects. The analyses showed a decrease of the energy dissipation rate for Yacyretá energy dissipator, with a hydraulic jump length that exceeds the length of the stilling basin, aspects that demand a continuous supervision of the works. On the other hand, the evaluation carried out for Jorge Cepernic dam does not exhibit significant changes of the hydrodynamic behavior of the stilling basin, except in a region downstream of the deflectors.*

Keywords: *Physical modeling, stilling basins, total dissolved gases.*

Paper ID: 1448

Numerical and Experimental Study on Energy Dissipation in Hydraulic Jump: A Comparison between Horizontal and Sloping Rough Channel Bed

Koroungamba Laishram[1], Thiyam Tamphasana Devi[1*], Ngangbam Romeji Singh[1] & Potsangbam Albino Kumar[1]
[1]*Department of Civil Engineering, National Institute of Technology, Manipur, Langol-795004, India*
E-mail: thiyam85@gmail.com (Corresponding address)

Abstract: *Hydraulic jumps are used as an energy dissipater, allowing excess energy from downstream flow to be dissipated through hydraulic structures, preventing scouring. Channel roughness is also necessary for good flow control, as it allows more excess energy to be dissipated downstream. To better understand the impact of channel roughness on dissipating energy in an open channel, this study investigates and compares the energy dissipation of hydraulic jumps based on roughened (gravel size 20mm) horizontal and sloping channel bed condition. The experiment was carried out in the Hydraulics Laboratory, Department of Civil Engineering, NIT Manipur (India) in a rectangular channel flume (16 m length, 0.6 m wide, and 0.8 m deep). The characteristics of hydraulic jumps (length of jump, upstream and downstream head, velocities, and subsequent depth ratio) were measured and using these characteristics, the amounts of energy dissipation are calculated. Results show that the sloping channel bed is much safer for the structure, dissipates more energy, and the control of the flow is much easier as compared to the horizontal condition. Thus, it can be suggested that sloping channel bed is an effective channel bed condition when energy dissipation is the main purpose. The experimental results were numerically simulated using CFD (Computational Fluid Dynamics) techniques and found to be within an acceptable range.*

Keywords: *CFD, Open channel flow, Hydraulic jumps, Bed roughness, Sloping channel, Energy dissipation.*

Paper ID: 1496

Investigation of energy attenuation, flow resistance and impending motion of downstream bed material in rock ramps

Vishal Singh Rawat[1], Thendiyath Roshni[1], Michele Palermo[2], Simone Pagliara[3] & Deep Roy[4]
[1]*Department of Civil Engineering - National Institute of Technology, Patna, India*
[2]*Department of Energy, Systems, Territory and Construction Engineering, University of Pisa, Pisa, Italy*
[3]*Laboratory of Hydraulics, Hydrology and Glaciology (VAW), ETH Zurich, Zurich, Switzerland.*
[4]*Department of Civil Engineering – National Institute of Technology, Tiruchirappalli, India,*
E-mail: roshni@nitp.ac.in

Abstract: *Block ramps are ecofriendly drop structures, which ensure stable downstream river bed, peculiar to flows over macro roughness elements. It uniquely serves an essential paradigm in riverine management, to encounter deliveries in an ecologically sound manner. It permits safe fish passage, stabilizes stream banks and bed profiles and creates habitat diversity. Study of flows over block ramps are quite extensive and are associated with many intricacies. In this paper, flume experiments were conducted in the hydraulic laboratory of National Institute of Technology Patna. The particle densimetric Froude number (F^*) was calculated for the mobile bed and chosen to predict the hydraulic conditions for incipient sediment motion in the uniform beds and it is compared with the established literature formulations to estimate the stability of the beds under large-scale roughness conditions. Scour volumes from each experimental run is quantified and the intensity of sediment motion was determined. Experimental data analysis allowed describing the mechanism of incipient motion of the mobile bed in the stilling basin, energy dissipation, flow resistance and in the interstitial flow over rock ramps and intensity of block movement for various flow regimes. It was found that the dimensionless*

shields stress increases with ramp slope and the intensity of ramp motion decreases with the shear stress. The results indicated that friction resistance increases with relative submergence for the tested range of experiments. A comparison with previous literature with uniform sediment transport indicates that relative roughness of block is responsible for increasing the dimensionless Shields stress. The results of physical testing can be used to assess and predict the effective dissipation of energy and its impact on the stability of rock structures.

Keywords: *Densimetric Froude number, energy dissipation, frictional resistance, incipient motion, intensity, rock ramps.*

Paper ID: 1431

Energy dissipation and flow regime downstream of trapezoidal piano key weirs

X. Shen[1] and M. Oertel[2]
[1,2]*Hydraulic Engineering Section, Helmut-Schmidt-University, Hamburg, Germany*
[1]*E-Mail: xiaoyang.shen@hsu-hh.de*
[2]*E-Mail: mario.oertel@hsu-hh.de*

Abstract*: In the past two decades, increasing numbers of piano key weirs (PKW) have been constructed due to their sufficient hydraulic performance and optimized space requirements. Extensive experiments and numerical studies have been carried out to identify the optimal geometry and maximize the discharge capacity. However, less attention has been paid to address energy dissipation processes and the flow structure downstream, especially for in-channel applications. Within the present study, symmetrical trapezoidal piano key weir models were fabricated in two model sizes with different cycle numbers via a 3D printing technique. Both models were tested under equivalent hydraulic conditions to investigate the potential influence of the cycle number and model size regarding energy dissipation processes and the flow structure downstream of the weir. Data show identical downstream residual energy for both investigated weirs, indicating that a model with 30 cm weir height and single cycle is able to deliver reasonable results in terms of total energy dissipation. However, the flow structure downstream of the weir was observed to differ with actual weir size and cycle number, indicating that small-scaled models with an insufficient cycle number may not be able to represent a similar outflow characteristic as the prototype. Sufficient cycle number was found to be beneficial for a more stable downstream flow condition. To facilitate practical design of geometrically similar weirs, observational insights of downstream flow condition development have been provided and analytical equations have been proposed for localization of aerated flow region as well as residual energy estimation.*

Keywords: *Cycle number, Energy dissipation, Flow characteristics, Piano key weir, Scale effect*

Paper ID: 1480

Experimental and Numerical Investigation on Critical Submergence for Square Water Intakes

B. Das[1], Z. Ahmad[1] & P. K. Sharma[1]
[1]*Department of Civil Engineering, Indian Institute of Technology Roorkee, Roorkee, India*
E-mail: bdas@ce.iitr.ac.in

Abstract*: Critical submergence for square water intakes in an open channel flow has been investigated experimentally and numerically in this paper. The depth of water at which the tail of a free surface vortex core just reaches the tip of an intake causing air entrainment is referred to as critical submergence. The formation of an air-entraining vortex in the vicinity of an intake is considered to be a severe problem for pumps. Experiments were conducted in a concrete flume of 9.47 m length, 0.5 m*

wide, and 0.6 m deep with an intake of size 0.04 m×0.04 m under uniform flow for different flow conditions. A three-dimensional Multiphase CFD Model was also developed for simulating critical submergence for the intakes. SST k-ω turbulence model together with the volume of fluid (VOF) two-phase (water-air) model were used to simulate the flow field in the water intake system. In the CFD simulation study, critical submergence was identified using phase volume fraction analysis and free surface streamline analysis. From both experimental and numerical studies, it is found that the approach flow Froude number and intake flow Froude number play an important role for estimating critical submergence depth. The critical submergence increases with an increase in intake Froude number and decreases with an increase in approach Froude number for a square Intake. The outcomes of the CFD simulation were validated with experimental data.

Keywords: *Critical Submergence, Square Water Intakes, Open Channel Flow, Vortex Formation, CFD, ANSYS FLUENT.*

Paper ID: 1478

Anti vortex device to operate pump intakes below the minimum submergence

M.A. Angulo[1], A. Rivetti[1], C.V. Lucino[1] & S.O. Liscia[1]
[1]*Laboratory of Hydromechanics, UNLP, La Plata, Argentina*
E-mail: mauricio.angulo@ing.unlp.edu.ar

Abstract: *Thermal power plants require water cooling pump stations to inject fresh water into the condenser. Generally, these pump stations have vertical mixed-flow pumps with large flows and low-to-medium heads that take water from rivers or reservoirs. Under certain natural conditions, water sources can reach extraordinarily low levels, thus affecting pump operation as the minimum recommended submergence might not be met, causing the power plant to be powered off. This causes both a very high loss of profit for the company and a decrease in the available power for the electrical supply system. Therefore, expanding the range of submergence under which pumps could be operated safely for special, transitory, and emergencies is an alternative worth exploring. Pump operation below the minimum level gives rise to unacceptable vortices on the free surface, air entrainment inside the circuit, higher mechanical vibrations, and lower efficiencies due to higher pre-swirl at the suction bell. Because of this, an anti-vortex device is proposed to prevent free surface vortex formation to allow the operation of the pump over a wider range of submergence. The device has a grating shape and was designed and optimized based on scale model tests performed on a typical pump bay following the ANSI standard intake design. Flow analyses through particle image velocimetry (PIV), acoustic Doppler velocimetry (ADV) velocity profiles, and swirl-meter measurements were carried out for a range of levels under the minimum and then compared with the device installed under the same conditions. Experiments proved the effectiveness of the solution to reduce free surface vortex formation and pre-swirl, allowing pumps to operate down to half of the recommended submergence.*

Keywords: *Pump station, submergence, free surface vortex.*

Paper ID: 1445

A Novel Dual Inlet Tangential Vortex Drop Shaft: An Analytical and Numerical Study

S. Mulligan[1] & S. Pope[2]
[1] *VorTech Water Solutions Ltd, Galway, Ireland*
[2] *Armatec Environmental Ltd, New Plymouth, New Zealand*
E-mail: sean.mulligan@vortechws.com

Abstract: *Urban drainage systems are undergoing substantial upgrades to enhance resilience to increased urbanisation, population growth and more intense rainfall events. Typically, in*

these hydraulic structures, the challenge of dropping water flows over significant elevation differences is overcome using vortex drop shaft structures. Many options are available for transferring flows from a single upstream channel to a lower level using independent vortex drop shafts. However, there are many cases where a manhole is to convey discharge from multiple flow inlets. In this work, a novel dual inlet tangential vortex drop shaft was developed during a full-scale commercial project proposal. The paper presents analytical and multiphase numerical modelling studies of various geometries of the structure. Analytical methods were used to derive conditions to maximize the free drainage discharge of inlets and were tested using a three-dimensional multiphase numerical model. The performance of the structure was evaluated for two drop tube diameters with scenarios including (1) equal flows in symmetric inlets, (2) unequal flows in asymmetric inlets and (3) independent inlet flows. The results demonstrate that the analytical model overestimates the free-drainage discharge. Nonetheless, interaction between flows and opposing inlets in the design did not appear to pose any significant stability or performance issues. Moreover, when tested at extreme flow rates up to flooding conditions, the numerical results demonstrate that the vortex air core remains intact. To conclude, a cost-benefit analysis of this type of configuration is presented which provides an economic argument for consideration of the dual inlet drop shaft in future schemes.

Keywords: *Energy Dissipation, Multiphase Modelling, Numerical Modelling, Urban Infrastructure, Vortex Drop Shafts.*

Paper ID: 1495

Submerged flow regimes downstream of a weir at multiple slopes

K.S. Vikas[1] & A. Sarkar[2]
[1]*Interdisciplinary Centre for Water Research (ICWaR), IISc Bangalore, India.*
[2]*School of Infrastructure (SIF), IIT Bhubaneswar, Khorda, India.*
E-mail: asarkar@iitbbs.ac.in

Abstract: *The higher water levels unanticipated at the time of floods are not generally considered in the design of hydraulic structures. The present study emphasizes the experimental investigation for the characterization of submerged flow regimes downstream of a rectangular sharp-crested weir at three different slopes (0, 0.004, and 0.008) that are possible with a laboratory flume. The classical flow patterns of submerged regimes, namely impinging jet, surface wave, surface jump, and surface jet regime, were analyzed. The experiments were conducted to examine the water profiles and the variations of different parameters, representing surface jumps and surface waves, with a gradual rise in submergence. The range of t/h values (where t and h are the downstream and upstream depths above the crest of the weir) at which these regimes shift from one to another were observed to vary with the slope. The photographs of the patterns of the submerged flow regimes at three different slopes and six variants of discharge were captured, and image analysis was carried out. The profiles were extracted for each submergence level to develop the correlations for various characteristics of surface jumps and waves. With an increase in the submergence, the non-dimensional lengths of the surface jumps decreased at all the slopes and were within a range of 10 to 55, whereas the non-dimensional heights increased and were within a range of 0 to 0.7. The surface wave with a maximum normalized amplitude of 0.6 and a maximum normalized wavelength of 4 faded out gradually as the regime ends.*

Keywords: Sharp-crested weir, Submergence, Flow regimes, Free Surface.

Paper ID: 1437

Prediction of Discharge Coefficient of Circular Side Orifice through Machine Learning Technique

Syed Ahmed Noman[1], Dr. Ajmal Hussain[2], Mr. Mohd. Danish[3] & Prof. Rashid Ali[4]
[1]*MTech (Artificial Intelligence), Aligarh Muslim University, India*
[2]*Assistant Professor, Department of Civil Engineering, Aligarh Muslim University, India*
[3]*Assistant Professor, Civil Engineering Section (University Polytechnic) Aligarh Muslim University, India*
[4]*Professor, Department of Computer Engineering, Aligarh Muslim University, India*
E-mail: syedahmednoman@zhcet.ac.in

Abstract*: A sharp-crested circular side orifice is a crucial element when it comes to diverting flow from primary source to its subordinate source. Such a flow measurement instrument technique is of immense value in conservation and evaluation of drainage and irrigation networks. Usually, it is placed towards the side of a channel in order to regulate the flow of the fluid. Traditionally, coefficient of discharge was predicted through regression methods which are time-consuming and lack accuracy. Artificial Intelligence (AI) and its applications in this domain have bridged this gap by providing novel alternative methods which prove much more efficient. Repeated studies have pointed out that AI techniques generally give better results when it comes to a myriad of water variables such as rainfall-runoff, evaporation and evapotranspiration, streamflow, and dam water level changes. Total 261 dataset has been collected from the literature review comprising of the fully submerged orifice and for partially-submerged orifice with varying orifice diameter (D) of 5 cm, 10 cm and 15 cm. This study aims to provide a better estimate of prediction of discharge through circular sharp-crested orifice using Artificial Neural Network (ANN). The ANN model has been deployed to randomly select 80% of the data for training, 15% for validation and remaining 5% for testing. In the ANN model, Lavenberg-Marquardt algorithm was used as back-propagation step to assign weights in order to predict the output. The correlation coefficient (R), mean absolute error (MAE) and root mean squared error (RMSE) for complete data of fully and partially submerged circular side orifice are observed to be 0.9765, 0.0228 and 0.0172 respectively.*

Keywords: *Artificial neural network, Circular side orifice, Coefficient of discharge, Flow measurement.*

Paper ID: 1440

Effect of deficient tailwater on the performance of slotted roller bucket of Sagar Dam, Madhya Pradesh

Vankayalapati S. Ramarao[1] Amit Kulhare[2]
[1&2] *Scientist -C, Central Water and Power Research Station, Khadakwasla, Pune-411 024. India*
Email: sramaraov@gmail.com

Abstract: *Slotted roller bucket type energy dissipators are provided for spillways when the tailwater depth is 1.2 times the sequent depth required for the formation of a hydraulic jump. In slotted roller buckets, the dissipation of energy occurs by the lateral spreading of jet passing through bucket slots in addition to the formation of two complementary rollers as in the solid bucket. These are provided for spillways where the downstream river bed is of sound rock. However, when the tailwater levels are not realised during the release of floods over the spillway and surface and ground rollers are not formed, it results in a possible sweep of the flood out of the bucket. Deficient tailwater levels during the initial period of operation of the bucket leading to ski action, generation of high hydrodynamic pressures during roller formation due to very high incoming velocities and occurrence of negative pressures on the bucket teeth, are some of the hydraulic parameters that lead to damage to the slotted roller bucket. Indira Sagar Dam is 653 m long and 92 m in height and was built on the River Narmada in Madhya Pradesh. The main and auxiliary spillway comprises 12 and 8 spans respectively of size each 20 m x*

17 m and is designed to dispose of a Probable Maximum Flood of 83,400 m^3/s. The energy dissipator provided was original in the form of a slotted roller bucket. The project is in operation from 2004-2005. Discharges of the magnitude 30,000 m^3/s were released during the monsoon of 2013 and then it was observed that the entire slotted roller bucket in front of spans 6 to 12 was washed away. The teeth of the roller bucket were overturned and thrown away by the flood waters. Theoretical analysis for actual discharges passed over the spillway from 2006 to 2012 indicated that the roller action was not forming for these conditions and instead ski action was taking place due to deficient tailwater levels. There was also a transition from ski action to roller action for increasing discharges till the tailwater build was realised. In fact, this is the inherent shortcoming of roller bucket type of energy dissipators and damages to the bucket have been reported elsewhere also. In this paper, the authors account for the causes that damaged the slotted roller bucket of Indira Sagar dam, M.P and recommendations suggested for improving the performance of energy dissipation arrangement.

***Keywords**: Slotted roller bucket, tailwater level, bucket teeth, apron, ground roller, surface roller, ski jump bucket*

Paper ID: 1473

Angle Effects of LIDAR Measurements on a Flat Surface and in High-velocity Spillway Flows

R. Li[1], K.D. Splinter[1], M. Kramer[2] & S. Felder[1]
[1] *Water Research Laboratory, School of Civil and Environmental Engineering, UNSW Sydney, Australia*
[2]*UNSW Canberra, School of Engineering and Information Technology (SEIT), Canberra, ACT 2610, Australia*
E-mail: s.felder@unsw.edu.au

***Abstract:** Measurements in aerated high-velocity spillway flows have been traditionally conducted with point-source instruments. LIDAR is a promising remote sensing technology that may be able to provide more detailed insights into the free-surface properties. So far only one laboratory study has investigated the use of LIDAR technology in air-water flows on a spillway model showing that free-surface data in the aerated part of the flow can be recorded with high spatial and temporal resolution. However, the spillway model, as well as the aerated flow region, were quite short and further research is needed to test the performance of LIDAR technology on a longer spillway. Herein, the present study tested a LIDAR in an 8.6 m long laboratory spillway with strip roughness. To ensure consistency of results along the spillway, the performance of the LIDAR was first assessed without flowing water along a straight wall as well as in the spillway model without water. The results showed that the LIDAR was affected by viewing angle that can be corrected using a scaling factor. The LIDAR was subsequently used to measure the free-surface properties of skimming flows. Comparative results for three different LIDAR measurement positions showed effects on the free-surface elevations and standard deviations linked with the different grazing angle of the LIDAR relative to the free-surface and with the waviness of the free-surface. A potential correction scheme for this is discussed.*

***Keywords:** Air-water flows, remote sensing, physical modelling, instrumentation, hydraulic structures, signal processing.*

Paper ID: 1427

Preliminary Evaluation and Design of a New Energy Dissipation Stilling Basin via Numerical and Experimental Modeling

F. Zabaleta[1], F.A. Bombardelli[1], S.A. Valbuena[1]
[1] *Department of Civil and Environmental Engineering, University of California, Davis, USA*
E-mail: fede.zabaleta@gmail.com

Abstract: *Design of stilling basins for the dissipation of energy has been established by the seminal tests developed by the U.S. Bureau of Reclamation and the Saint Anthony Falls Laboratory, by mid XX century. Such designs basically include blocks and end sills, dentated or complete. However, those designs need variations in desertic zones, where the end sill is not allowed due to self-cleaning purposes, and no "drop" is possible for the same reason. In the case of the outlet structure for Perris Dam, located in the Riverside County, California, the design of the stilling basin was optimized using a Computational-Fluid-Dynamics (CFD) approach, and a physical model. Potential overspill due to unsteady oscillations of the free surface and energy dissipation were studied using a Volume of Fluid (VoF) method combined with a RANS turbulence closure, and then validated with a physical model. The combination of powerful numerical simulations with experimental models allows for an efficient analysis and design of complex hydraulic structures. The cheaper numerical simulations provide an accurate description of the flow that can be used to test several geometries and evaluate the most suited one to be represented on the physical model. From this first stage, an optimized geometry is implemented in a physical model which is used to calibrate and validate the numerical simulations as well as to optimize the geometries obtained from the numerical model. The new proposed structure promoted a high level of dissipation and a design that reduce the maintenance issues associated with desertic regions.*

Keywords: *Energy dissipator, hydraulic jump, numerical modeling, physical modeling.*

Paper ID: 1456

Stilling Basins Using Wedge-Shaped Baffle Blocks

N.N. Pillai[1] & M.L. Kansal[2]
[1]*Emeritus, School of Engineering, Amrita University, Coimbatore, India*
[2]*Water Resource Development and Management, IIT Roorkee, Roorkee, 247667, India*
E-mail: mlk@wr.iitr.ac.in

Abstract. *The United States Bureau of Reclamation (USBR) extensively studied hydraulic jump-type stilling basins for various inflow and tailwater conditions. They recommend Basin III using chute blocks, rectangular baffle blocks, and sloping end sill for medium to high dams. However, a significant contribution to the knowledge regarding the production of turbulence in discontinuity layers, the dissipation of energy, the convection, and the decay of turbulence in hydraulic jump was made by subsequent researchers. However, while developing the stilling basin, the wake formed by the jets separating at the front corners of the rectangular baffle blocks in Basin III and reattaching the downstream of the blocks was not considered. The use of Basin III was found limited to lower dams since cavitation of the baffle blocks was observed in many dams. After that, the use of wedge-shaped baffle blocks below a model spillway was systematically investigated. It was noticed that baffle blocks with a vertex angle of 120^0 cut back at 90^0; the separating jets do not reattach downstream. Due to the wedge shape (super-cavitating), the low-pressure cavities do not collapse on the boundary. The new forms of baffle blocks are found to offer more drag to the flow when compared to rectangular blocks. Model tests were conducted to evolve stilling basins using the new baffle blocks and were highly successful. For example, in a stilling basin with an Inflow Froude Number of 7.07, with new blocks, a floor-length of about half the earlier suggested length is found to have better characteristics of scouring and sweep-out depth as compared to USBR Stilling Basin III.*

Keywords: *Wedge-shaped Baffle Blocks, Stilling Basins, hydraulic jump*

Paper ID: 1469

Physical and Numerical model studies for Hydraulic design of Stilling basin as an Energy dissipator of a Spillway - A Case study

B.S. Sundarlal[1], Mrs. Vaishali Gadhe[1], Mrs. S.R. Patnaik[1], Amol H. Gaikwad[1]
[1]*Central Water and Power Research Station, Pune -411024, India*
E-mail: srinivasaslal@gmail.com

Abstract : *The design of the stilling basin for a dam spillway is site specific and hydraulic model studies are required to be carried out for evolving the best possible design. A properly designed hydraulic jump stilling basin can ensure 60-70% dissipation of energy in the basin itself. Supported by hydraulic model studies and prototype experience, a suitable stilling basin caters to a variety of conditions covering discharge intensity, head, tail water variation, and topographical conditions. The most serious problem with hydraulic jump type dissipator is more structural strength rather than hydraulic efficiency. Experience in recent years gives many examples of stilling basins having damages due to uplift, vibration, cavitation and abrasion, all having their origin in the internal structure of hydraulic jump. Model studies play an important role in optimizing the length of the stilling basin, elevation of the stilling basin floor level for a high head spillway and height of the end sill and hydraulic jump characteristics. This paper discusses hydraulic model studies conducted at Central water and Power Research Station (CWPRS), Pune, India, for the hydraulic jump stilling basin as energy dissipator for Kotlibhel dam spillway stage 1-B, Uttarakhand, India, at the conceptual stage, which played an important role in enhancing the overall performance of spillway and energy dissipator by incorporating modifications such as lowering of stilling basin floor level and modifying the height of the end sill. Through recent advances in computing power and modeling software capabilities, it is now feasible to undertake complex three-dimensional analysis using Numerical/Computational Fluid Dynamics (CFD) modeling techniques. Numerical model studies were carried out to assess the performance of the stilling basin using Computational fluid dynamics software, FLOW-3D. The major benefit of the CFD modeling is that it allows early identification of problematic flow features and modifications to the design/layout could be tried rapidly and cost-effectively. The results obtained from the numerical model were compared with the results of the physical model studies. Comparison of the results from the two modeling approaches has provided a very good insight into the design process to optimize the stilling basin design. This use of physical and Numerical modeling (CFD modeling) techniques has provided invaluable insight and greater confidence for future use of standalone CFD analysis in spillway/energy dissipator design as complementary tool for physical model studies. After studying the performance of the stilling basin for Kotlibhel stage 1-B Hydro electric project, both through the physical and numerical techniques, it is found that CFD modeling can be used as a valuable tool to test the preliminary empirical design and in the optimization phase of the stilling basin design prior to conducting physical model studies. The details of physical and numerical model studies conducted for improving the overall performance of stilling basin are described in the paper.*

Keywords: *Hydraulic jump, Numerical modeling physical modelling, Spillway, Stilling basin, Tailwater level.*

Paper ID: 1471

Local Scour Downstream of Type-A Trapezoidal Piano Key Weir

Ch. A. Chooplou[1], A. Ghafouri[1], M. Ghodsian[1*] & M. Vaghefi[2]
[1]*Department of Civil and Environmental Engineering, Tarbiat Modares University, Tehran, Iran*
[2]*Department of Civil Engineering, Persian Gulf University, Bushehr, Iran*
[*]*E-mail : ghods@modares.ac.ir*

Abstract*: A major issue with weirs over erodible beds is the scour downstream, which can seriously challenge their stability. This paper presents experimental results on trapezoidal type-A piano key weirs having two values of bottom slopes keys (28° and 47°). Experiments were conducted in a recirculating channel 10 m long, 0.75 m wide and 0.8 m high. Approach flow depths in the range of 0.032 to 0.07 m, and tailwater depths in the range of 0.08 to 0.18 m were evaluated. It was observed that the bottom slope of weir keys significantly influences the scour depth, such that a 67.8% increase in the slope, increased the average maximum scour depth by about 65%. The maximum scour depth occurred downstream of the outlet keys. Moreover, the scour increased with an increase in the approach flow depth and decreased with the tailwater depth. Distance of location of the maximum scour depth from the weir toe, and length of the scour hole in the weir with key bottom slope of 47° were respectively 5 and 21% greater than the weir with key bottom slope of 28°. In general, the geometric dimensions of the scour hole which were influenced by tailwater depth were more obvious in the model with smaller key bottom slope, such that with a 125% increase in the tailwater depth, the average variations of the scour parameters (maximum scour depth, its distance from the weir toe, and the scour hole length) in the model with smaller key bottom slope were respectively 1.5, 2.5 and 2.7 times those of the other model.*

Keywords: *Experimental Study, Scour, Piano Key Weir, Weir Geometry, Key Bed Slope*

Paper ID: 1483

Hydraulic Performance of Piano Key Weir with Linear and Curvilinear Profiles of Outlet Keys

Prakash Kumar Sinha[1], Binit Kumar[2] & Zulfequar Ahmad[3]
[1]*Research Scholar, Indian Institute of Technology Roorkee, Roorkee 247667, India*
[2]*Former PhD, Indian Institute of Technology Roorkee, Roorkee 247667, India*
[3]*Professor, Indian Institute of Technology Roorkee, Roorkee 247667, India*
E-mail:pk_sinha@ce.iitr.ac.in (Corresponding Author)

Abstract *With rising demand for storing water in reservoirs, many spillways require modification by optimizing their shape for enhancing their discharge capacity. A Piano Key Weir (PK weir) is a modified weir and it is installed at the dams to augment the discharge capacity of the weir. Due to different geometrical configurations of PK weirs, the hydraulic performance of the weir is still in evolving research. In this study, hydraulic performance of two different models of PK weir is studied experimentally. All the configurations of both the PK weir are same except the shape of the outlet keys. The outlet key of the first model is linear whereas for the second model a curvilinear ogee shape is adopted. Ten discharges between 0.0075-0.03 m^3/s and ratio of head over crest to weir height between 0.17-0.58 m were considered for each model. Water surface profiles were plotted along longitudinal and transverse directions at various sections of the PK weir. The effect of the weir geometry on discharging capacity was studied and it is found that the PK weir with curvilinear shaped outlet key is hydraulically more efficient than the linear shaped key. The coefficient of discharge of PK weir with curvilinear shaped outlet key is 12.41-18.15% more than the linear shaped key.*

Keywords: *Curvilinear, Hydraulic-performance, Piano Key Weir, Water flow profile*

Paper ID: 1489

Experimental Study on Three-Dimensional Velocity near Piano Key Weirs

Binit Kumar[1] & Zulfequar Ahmad[2]
[1]*Former PhD candidate, Department of Civil Engineering, Indian Institute of Technology Roorkee, Roorkee 247667, India*
[1]*Postdoctoral fellow, Department of Civil Engineering, National Institute of Technology Warangal, Warangal-506004, India*
[2]*Professor, Department of Civil Engineering, Indian Institute of Technology Roorkee, Roorkee 247667, India*
E-mail: bkumar@ce.iitr.ac.in (Corresponding Author)

Abstract: *A Piano key weir (PK weir) is a non-rectilinear spillway commonly used to augment the discharge capacity at the dams and diversion structures. Installation of such hydraulic structures may alter the flow characteristics, sediments transport and sedimentation in the river channels. The present study deals with an experimental investigation of three dimensional velocities near the two different laboratory models of PK weir having noses with an outlet overhangs. The purpose of this study is to observe the flow field in the vicinity of a one-cycle and two-cycle PK weir to understand the flow behavior of the weir that may be useful for sediment movement over weir. The velocity profile was measured at various grid points for both models considering unit discharges of 48.5×10^{-3} m^2/s and 49×10^{-3} m^2/s. The velocity fields near both inlet and outlet keys were examined and it was found that there is a considerable increase in the vertical (y-component) and lateral (z-component) velocities. It was observed that the rise in the mean velocity along longitudinal, vertical and lateral direction was more for two cycle PK weir due to increased crest length. For both discharge, the maximum velocity was observed near the inlet key. Computational Fluid Dynamics (CFD) simulations were also used to understand the flow configuration over the PK weir. It was seen that the magnitude of velocity increased due to vertical contraction near the weir and the flow velocity is very high immediately downstream of PK weir.*

Keywords: *CFD, lateral-direction, outlet key, Piano Key Weir, velocity.*

Paper ID: 1455

A Numerical Investigation on Residual Energy of Labyrinth Weirs

P. Langohr[1], B.M. Crookston[2] & D.B. Bung[1]
[1]*FH Aachen University of Applied Sciences, Aachen, Germany*
[2]*Utah Water Research Laboratory, Utah State University, Logan (UT), USA*
E-mail: langohr@fh-aachen.de

Abstract: *The replacement of existing classical weirs with labyrinth weirs is a proven techno-economical solution and a means to increase the discharge capacity when rehabilitating existing structures. However, flows exiting a labyrinth weir are complex, three-dimensional, and aerated; additional information is needed regarding energy dissipated by such weirs. In this study, three labyrinth weirs with different crest lengths were simulated with FLOW-3D HYDRO. Reynolds-averaged Navier-Stokes (RANS) modeling with the use of finite-volume method and Re-Normalisation Group (RNG) k-ε turbulence closure were employed. An attempt was made to more precisely account for the flow field in the downstream region. Consequently, the velocity head was determined with both depth-averaged and section-averaged velocities. Additionally, the kinetic energy correction coefficient was considered. A comparison of the computational fluid dynamics (CFD) results with prior experimental data showed that the residual energy was influenced by factors such as probe position and geometric parameters. A minor influence was observed for the kinetic energy correction coefficient. Overall, the high amount of energy dissipation was underlined and an acceptable agreement between simulated and literature data was documented.*

Keywords: *CFD, energy dissipation, labyrinth weirs*

Paper ID: 1458

Introduction and investigation of a large-scale piano key weir fabricated via rapid prototyping

M. Oertel[1], B. Belay[2], X. Shen[3] & H. Willems[4]
[1,2,3,4]*Hydraulic Engineering Section, Helmut-Schmidt-University, Hamburg, Germany*
[1]*E-mail: mario.oertel@hsu-hh.de*

Abstract: *Piano key weirs (PKW) are novel hydraulic structures, characterized by an increased discharge capacity, compared to regular weirs. Through the last decades, experimental studies have mainly focused on efficiency aspects and design regulations. Therefore, often small scaled experiments were conducted in hydraulics laboratories in which model geometries were produced with materials like acrylic glass. But also in recent practices, 3D printing techniques are being used to plot small-scaled piano key weir configurations up to 30 or 40 cm width. The present investigation focuses on a large-scale 3D printed PKW with a total width of 1 m featuring quarter round crest shape in upstream direction. The weight of the printed structure is approximately 30 kg and 8 days continuous plotting process were necessary to produce the weir – consequently, it is the largest 3D printed PKW up-to-date. The plotted PKW was installed at Helmut-Schmidt-University's new hydraulics laboratory within a 20 m long and 1 m wide flume. Discharges ranging from 20 to 330 L/s were used to investigate the hydraulic performance of this printed PKW with a height of 40 cm. Results depict that the scale difference between models of 30 cm and 40 cm in height is not significant if the objective is to study discharge coefficients. 30 cm models can reproduce similar flow properties at a relatively smaller cost and less preparation time. Moreover, patterns of nape detachment were observed at the downstream crest lip, indicating the influence of the printing layer height in inducing undesired flow separation for relative upstream heads above 0.33 that could likely affect the nape trajectory.*

Keywords: *Discharge coefficients, Large-scaled model, Piano key weir, 3D printing*

Paper ID: 1474

River aggradation-degradation under sand mining: experimental and numerical studies

Nejib H. Abdullahi[1] and Zulfequar Ahmad[1]
[1]*Indian Institute of Technology Roorkee, Roorkee, India*
E-mail: nejibhassen1@gmail.com

Abstract: *Sand mining from the river for different purposes is a common cause of the failure of hydraulic structures. Its presence in a river causes intense bedload transport, which is crucial for designing and operating hydraulic structures. The flow characteristics and aggradation-degradation phenomenon in the vicinity of the sand mining pit have been analyzed with experimental and numerical approaches. The bed level evolution in the vicinity of the pit was measured using a pointer gauge for different temporal and equilibrium stage conditions. A numerical simulation was done using a three-dimensional computational fluid dynamics model, Flow-3D, which was used to compute the bedload transport deposited in the pit and to estimate the spacetime dynamics of the bed evolution of the pit. The volume of fluid (VOF) equation is used to confirm the application of boundary conditions at the free surface in a numerical method. The turbulence formed in the vicinity of the pit was predicted using the turbulence model k-ε with re-normalization group (RNG) extensions. Different bedload transport equations and mesh grid sizes were tried to identify the best fit for aggradation-degradation simulation around sand mining pit. The total volume of bedload transported from the upstream section was deposited in a pit since a pit was used as a bedload trap. The experimental and numerical results showed a close agreement which verified the efficiency of the Flow-3D model.*

Keywords: *Sand mining, bedload transport, pit morphology, aggradation, degradation, bed evolution*

Paper ID: 1452

Effect of Diameter and Inlet-depth on Hydro-suction performance of a Suction pipe

A. Jaiswal[1], Z. Ahmad[1] & S.K. Mishra[1]
[1]*IIT Roorkee, Roorkee, India*
E-mail: ajaiswal@wr.iitr.ac.in

Abstract: *Sedimentation in rivers and reservoirs leads to inundation of surrounding areas, topsoil degradation, low depth for navigation, loss of reservoir capacity, etc. Hydro-suction is a process of sediment removal in which sediment is sucked along with water using a suction pipe placed vertically on/above/below the sediment bed. This paper deals with the effect of diameter and inlet-depth of suction pipe on performance of hydro-suction. A series of experiments are performed using five suction pipes of diameter of 5.08×10^{-2} m, 7.62×10^{-2} m, 10.16×10^{-2} m, 12.70×10^{-2} m, and 15.24×10^{-2} m, placed at inlet depth of 0.015 m and 0.03 m, under discharge ranging from 0.5×10^{-3} m^3/s to 3×10^{-3} m^3/s and median sediment size of 0.33 mm. Hydro-suction performance is evaluated by the sediment volume removed, which is calculated from the experimental data of scour profile. The investigation inferred that for a constant diameter, hydro-suction performance decreases with an increase in suction inlet depth. An increase in hydro-suction performance is seen up to suction pipe diameter of 10.16×10^{-2} m, and any further increase in suction pipe diameter decreases the hydro-suction performance.*

Keywords: *Hydro-suction, Suction pipe diameter, Suction inlet-depth, Scour profile, Scour volume.*

Paper ID: 1498

Bed Load Computation For Gravel Particles In Gravel-Silt Mixture

U.K. Singh[1], S. Kumar[1] & Z. Ahmad[2]
[1] *Koneru Lakshmaiah Education Foundation, Vaddeswaram, India*
[2] *Indian Institute of Technology Roorkee, Roorkee, India*
E-mail: umesh.ais@gmail.com

Abstract: *Fluvial hydraulics deals with the process of erosion, transportation, and deposition of sediment in channels by the action of flowing water. This paper presents experimental results on bed load transport rate of gravel particles transported from mobile channel bed made of gravel-silt mixture. Gravel and sand particles were observed transported as bed load due to their coarser size while silt transported as suspended load. Bed load was collected at regular time interval of like 15 minutes, 30 minutes, 60 minutes, etc. Bed surface profile and water surface profile were measured spatially at an interval of 50 cm at the center line of the test section along the flow. It was observed that collected bed load decreases with the time passes and the run was continued till the equilibrium condition i.e., low amount of bed load collected for a long duration and simultaneously bed surface profile and water surface profile come into nearly stable state. This condition is here treated as equilibrium condition as there is no feeding of sediment in the channel and test section also tends to in stable condition with nearly no outflow of sediment. At the end of run, layer of gravel particles was found on the top surface of the channel bed in the mixture of gravel-silt; however, high degradation was observed towards the upstream of test section. A relationship has been developed to compute the total bed load which found in good agreement with the observed data.*

Keywords: *Bed load, Cohesionless sediment, Equilibrium condition, Gravel particles, Hydraulics, Sediment transport.*

Paper ID: 1457

2-D simulation of flow structures over dunes for flow characteristics estimation

Mohammad Sharifi[1], Mohammad Reza Majdzadeh Tabatabai[2], Seyed Hossein Ghoreishi Najafabadi[2]
[1]*PhD Candidate, Department of Civil, Water and Environmental Engineering, Shahid Beheshti University, Tehran, Iran*
[2]*Assistant Professor, Department of Civil, Water and Environmental Engineering, Shahid Beheshti University, Tehran, Iran,*
E-mail: m_majdzadeh@sbu.ac.ir

Abstract: *Rivers have undulated beds, which are called bedforms. Depending on the hydraulic conditions, the bedforms have different types that cause resistance in the flow. Despite various research and experiments on bedforms, the topic is still debatable and requires further research. The present study is mainly focused on simulating the flow motion numerically on dunes in open channels to evaluate the effect of dune geometry on flow structure. Twenty-nine simulations were conducted to study the effect of the geometry of five types of dunes with different angles and heights under different hydraulic conditions and bed roughnesses. RANS and DES turbulence models were used to simulate small and large-scale dunes, respectively. The results of the numerical model were compared with the experimental results of previous researchers to validate the work, indicating the appropriate accuracy of the numerical model. Then, an empirical equation was adopted to evaluate the effect of dune geometry on flow hydraulics by considering the equations of previous researchers. Finally, sensitivity analyses were carried out to determine the dependence of each parameter in this equation.*

Keywords: *Bedform, Dune, Numerical simulation, Turbulence models.*

Paper ID: 1500

Predicting Riverbed Deformation During Floods for a Sustainable Design of Headworks Structures

I. Joshi[1,2], U. Singh[2], M. B. Bishwakarma[2], Y. Kitamura[3] & L. Lia[1]
[1]*Norwegian University of Science and Technology, Trondheim, Norway*
[2]*Hydro Lab, Lalitpur, Nepal*
[3]*Chigasaki Research Institute, Chigasaki, Japan*
E-mail: ishwar.joshi@ntnu.no

Abstract*: Understanding and predicting riverbed deformation during floods is important for the proper and sustainable design of headworks structures. Field-scale study of floods is challenging, and existing literature shows that mainly qualitative data (Photographic and Videographic) have been used in riverbed deformation studies. On the other hand, laboratory experiments allow studying floods in more detail and under a controlled environment. In this research, bed deformation of river channel during floods was investigated by conducting experiments in a curved flume. Parameters of the experiments were chosen with respect to gravel-cobbled River in Nepal. Four different floods with different magnitudes and durations were investigated in the experiments. Riverbed deformation was quantified by differencing pre and post-flood DEMs obtained using Structure from Motion (SfM) technology. The maximum scour during the floods was measured by scour chains made with light beads in a thread. The results showed that the magnitude and duration of floods are both important factors for riverbed deformation. The volume of channel bed erosion increased with the increase in flood magnitude. However, the total volume of sediment reworked was higher in the case of floods which are of lower magnitude but occur frequently in multiple cycles. The depth of maximum bed scour during the floods was observed much higher than the channel bed resulted at the end of the floods. The difference was more prominent along the bends where the secondary currents are stronger. The understanding could be used to better validate sediment transport models, and thus predict riverbed deformation more accurately for sustainable design of headworks structures.*

Keywords: *Floods, Maximum scour, Sediment transport, Numerical modeling.*

Paper ID: 1482

CFD-Based Modeling for Computing Discharge Coefficient of an Ogee Spillway

Hariom Gautam[1], Zulfequar Ahmad[1], & Pramod Kumar Sharma[1]
[1]*Indian Institute of Technology, Roorkee, India*
E-mail: h_gautam@ce.iitr.ac.in

Abstract: *The basic functionality of the spillway is to control the flow and safely convey the excess flow while keeping the structure stable. An ogee spillway is commonly used due to its high discharging capacity. The discharge coefficient of an ogee spillway is greatly affected by the ratio of an operating head and design head. Since physical models are expensive and have limitations in terms of scale effect, a comprehensive numerical simulation was performed in this study to estimate the discharge coefficient for an ogee spillway under various head ratios using CFD-based numerical modeling. The numerical simulations were done using FLOW-3D. In the present study, renormalized group (RNG) $k-\varepsilon$ turbulence closure model and the Volume of Fluid (VOF) algorithm were used for the simulation of flow over an ogee spillway. The discharge coefficient of the ogee spillway computed using CFD simulation for various head ratios shows a reasonable agreement with the values reported in the literature. Also, the water surface profiles, pressure distribution, discharge rating curve, and velocity distribution over the ogee spillway for the different heads were plotted and analyzed.*

Keywords: *Ogee spillway, Numerical modeling, CFD, Discharge coefficient, FLOW-3D.*

Paper ID: 1454

Effect of sinuosity variation on flow characteristics of sand mined sinuous channel using numerical modeling

O.P. Maurya[1], K.K. Nandi[1], S. Modalavalasa[1] & S. Dutta[1]
[1]*Indian Institute of Technology, Guwahati, India*
E-mail: om1996@iitg.ac.in

Abstract: *Anthropogenic activities within the river corridor can have a long-term influence on river health if not managed effectively. Uncontrolled practices of sand mining, a common but aggressive activity can cause severe river bed degradation and, as a result, river incisions, lateral channel instability, and bed armoring. This can further affect the downstream structure such as the bridge pier as well as the morphology of the river. The purpose of this study is to examine, with the help of a numerical model, the impact of a mining pit on the downstream pier and the main channel hydrodynamics if it is present on the flood plain of a sinuous channel. To carry out the study we have considered a meandering channel with two different sinuousness (1.1, 1.25). Additionally, a rectangular mining pit and a downstream circular bridge pier were considered on the flood plains of the channel. The numerical model used for the study was Flow 3D HYDRO with the renormalized group turbulence solver (RNG). As sinuosity increases, the highest zone of streamwise velocity narrows near the mining pit and widens out just downstream of the bridge pier, according to the findings. Near the sand mining pit, the shear layer zone shifted towards the main channel from the near bank. The secondary current at the outer bank near the mining pit is more concentrated in both cases. The channel having more sinuosity has shown a significant change in turbulence kinetic energy (TKE) as compared to the other channel. Additionally, this study can provide insight into the morphodynamics of meandering rivers under different in-channel disturbance conditions and thus be helpful in the proper planning and management of river health.*

Keywords: *(Sinuous channel, Floodplain, Sand mining pit, Bridge pier, Flow-3D, Flow characteristics).*

Paper ID: 1451

Estimation of Discharging Capacity and assessing the performance of Overflow Spillway with Full and Partial Operation of Gates using Numerical Modelling

Dr. Prajakta P. Gadge[1]& Kunal Kapur[2]
[1]*Scientist C, Central Water and Power Research Station, Pune, India*
[2]*Research Assistant, Central Water and Power Research Station, Pune, India*
E-mail: prajaktagadge@gmail.com

Abstract: *The spillway is among the most important structures of a dam project. The operation of spillway gates during the flood and fulfilling the downstream requirement is one of the main problems in reservoir management. Generally, the equal opening of all the gates during spillway operation is mostly preferred for maintaining an equal distribution of discharge in the energy dissipator and at the downstream side. A systematic study is required to determine the ability of the spillway to pass the discharge at various reservoir water levels through a specific opening of gates and assess the performance for the safety of the structure. The design of each project is unique and site-specific. A physical model is an indispensable tool to optimise the hydraulically efficient and economical design of spillways. However, the evaluation of spillway performance especially for the partial operation of gates is a time-consuming task on the physical model. Nowadays, the computational fluid dynamics technique is becoming popular in modelling spillway flows. This paper discusses the studies carried out for overflow spillway for gated and ungated operations. The Computational fluid dynamics software FLOW-3D was used for numerical simulation. The numerical model was validated by comparing the results with the physical model for ungated operations. The results in terms of design discharge passed over the spillway and corresponding pressures over the spillway surface computed using a numerical model were found closer to the results obtained from the physical model. Numerical model studies were extended further for the gated operation. Large data was generated which was used to create discharging capacity plots and evaluate the performance of the spillway for operating the spillway with different openings of gates at various reservoir water levels.*

Keywords*: Overflow spillway, spillway gates, physical model, numerical model, discharging capacity.*

Paper ID: 1426

Numerical Simulation of Air Entrainment in Stepped Spillways

F. Zabaleta[1], S. Márquez Damián[2], F.A. Bombardelli[1]
[1]*Department of Civil and Environmental Engineering, University of California, Davis, USA*
[2]*Research Center for Computational Methods (CIMEC), Santa Fe, Argentina*
E-mail: fede.zabaleta@gmail.com

Abstract: *Existing methods to simulate air entrainment in stepped spillways are based on a sub-grid air-entrainment function that accounts for the incorporation of air through the air-water interface; a transport equation for the bubbly phase is then solved inside the water domain. Unfortunately, these techniques are unable to represent bulking, i.e., the increase in water depths after the inception point. In this work, we introduce a novel method for the simulation of flow in stepped spillways (including the aerated region). The new air-entrainment model is implemented in the OpenFOAM platform and couples an air-concentration-transport equation with the traditional Volume-of-Fluid (VoF) method to account for bulking. The model was calibrated and validated using the experimental data by Amador (2005) and Felder and Chanson (2013). Air concentration and mean free surface location in the aerated region (defined as where the concentration of air is 90%) are analyzed on the top of the steps as well as inside the cavities. Very satisfactory agreement in the wall-normal distribution of air concentrations between model and experiments was obtained. The model showed to be capable of predicting bulking in stepped spillways, without the need for ad-hoc treatments*

Keywords: *Air entrainment, numerical modeling, stepped spillways.*

Paper ID: 1492

SPH Simulation of Non-Aerated Flow over Smooth Invert, Converging Spillways

J.D. Nóbrega[1], J. Matos[2], H.E. Schulz[3] & R.B. Canelas[4]
[1]*Federal University of Goiás, Goiânia, Brazil*
[2]*CERIS, Instituto Superior Técnico, Universidade de Lisboa, Lisbon, Portugal*
[3]*School of Engineering of São Carlos, University of São Paulo, São Paulo, Brazil. Hydro Engineering Solutions, Auburn, Alabama, United States*
[4]*Bentley Systems, Lisbon, Portugal*
E-mail: juliana.nobrega@ufg.br

Abstract: *Spillways may require converging sidewalls because of site or economy constraints. The adverse effects of the converging walls on supercritical flows include the formation of standing waves. In this context, numerical simulations were carried out using the Smoothed Particle Hydrodynamics (SPH) method. The numerical results were compared with experimental data acquired on a spillway model composed of a broad crested weir followed by a 1V:2H sloping chute with wall convergence angle of 9.9º or 19.3º, for a range of discharges. The flow depths at the spillway centerline were reasonably well predicted by the numerical simulations, except on the transition from the weir to the chute. Experimental data and numerical results showed that the sidewall flow depth normalized by the centerline flow depth (uninfluenced by the converging wall), tended to increase along the spillway, reaching maximum values of approximately 2 and 5, for wall convergence angles of 9.9° and 19.3°, respectively. Numerical cross-sectional flow depth profiles were also obtained, showing distinct shapes of the standing waves, depending on the wall convergence angle and position along the spillway. Overall, the simulated development of the standing wave width compared well with the experimental counterparts. The experimental velocity profiles obtained at the spillway centerline with wall convergence angle of 9.9º were also compared with those obtained numerically and the results revealed a fairly good prediction, except near the upstream end of the chute and close to the invert.*

Keywords: *Smoothed Particle Hydrodynamics, spillway, standing wave, supercritical flow, wall convergence.*

Paper ID: 1461

Jet Scour in Cohesive and Non-Cohesive Soil

J. Di Nardi[1], M. Palermo[1], S. Pagliara[1] & F.A. Bombardelli[2]
[1]*Department of Energy, Systems, Territory and Construction Engineering, University of Pisa, Italy*
[2]*Department of Civil and Environmental Engineering, University of California, Davis (CA), USA*
E-mail: s.pagliara@ing.unipi.it

Abstract: *A safe design of hydraulic structures depends on reliable estimations of the scour generated by the interaction between flow, structure, and bed sediments. Non-cohesive and cohesive soils are characterized by very different scour processes. Consequently, the behavior of the two soil types under scour has usually been studied separately. More specifically, cohesive soils are subjected to significant interparticle forces, which affect the properties of the material, resulting in different erosion mechanisms. Because of the complexity of the problem, erosion processes have been mainly investigated experimentally, resulting in a variety of case-dependent predicting equations. Recent theoretical advancements contributed to clarify the physics of scour in granular materials, paving the way to the development of general tools for its assessment. Nevertheless, a systematic characterization of local erosion in different types of soil is still lacking. The aim of this paper is to present an updated summary of current knowledge on scour caused by circular jets for cohesive and non-cohesive soils, highlighting differences and similarities of the scour phenomenon. In so doing, we also point out the still-challenging aspects and the future lines of research.*

Keywords: *Cohesive soil, granular soil, hydraulic structures, scour, turbulent jet.*

Paper ID: 1481

Scour downstream of a Broad Crested drowned Weir

Rajat Chauhan[1], Rakesh Kumar Chaudhary[2] & Zulfequar Ahmad[1]
[1]*Department of Civil Engineering, Indian Institute of Technology Roorkee, Roorkee, India*
[2] *Institute for Integrated Hydro Environment Research, Kathmandu, Nepal*
E-mail: corresponding, logtokumarrakesh063@gmail.com

Abstract*: A detrimental problem of scour downstream of a Bagmari Syphon in the feeder canal off taking from Farakka Barrage on the Ganga River, West Bengal, India has kindled our interest in understanding the scour process downstream of a broad crested drowned weir. A river was syphoned across the Feeder canal at Bagmari, flushing the top of the syphon with canal bed. However, the canal bed degraded over the years and resulted in exposure and protrusion of the syphon barrel above the canal bed and acting as a broad crested drowned weir. Due to the acceleration of flow over the barrel, scour has developed downstream of the barrel. This problem has been formulated and studied in the laboratory to study the scour process and its mitigation measures using boulders. Experiments have been carried out in the Hydraulics Laboratory of IIT Roorkee, India in a flume of 12.5 m length, 0.46 m width, 0.70 m depth for the different protruding height of weir and flow intensity, keeping ratio of shear stress of approach flow to critical shear stress less than unity. Study on factors affecting maximum scour depth and its temporal variation were analyzed graphically. The scour depth increases with an increase in protruding height of weir and flow intensity. It has been found that maximum scour reduces significantly using boulder apron. The maximum reduction in scour depth with boulder apron lies in the 44% - 68% range and the range of shift in maximum scour depth from weir with boulder apron was 1.6-1.8 times from the original location of maximum scour depth without boulder apron.*

Keywords: *Broad crested weir, drowned, flow intensity, riprap, scour*

Paper ID: 1494

Preliminary Analysis of The Effect Of Worked Wood Piles In Straight Channels

Deep Roy[1] & Simone Pagliara[2]
[1]*Department of Civil Engineering – National Institute of Technology Tiruchirappalli, Tiruchirappalli, India*
[2]*Laboratory of Hydraulics, Hydrology and Glaciology (VAW) – ETH Zurich, Zurich, Switzerland*
E-mail: pagliara@vaw.baug.ethz.ch

Abstract: *Eco-friendly low-head structures such as blunt nosed chevrons, wood bundles, worked wood piles etc. provide an effective solution to ensure grade and sediment load control in rivers, as well as to introduce morphological variations in the channel bed promoting biodiversity. Worked wood piles are usually installed near the outer bank in order to redirect the flow away from it, thus stabilizing the basin. Moreover, these structures increase the natural organic content in channels and create pronounced scour and dune features in their vicinity, giving shelter to a host of aquatic organisms and fish species. However, till date, a comprehensive analysis of the scour features occurring in the presence of worked wood piles is absent in literature. To shed light on this underexplored topic, the present study provides some initial results on the equilibrium and temporal scour characteristics in the vicinity of worked wood piles. Experiments were conducted with such structures in straight channels under a wide range of hydraulic conditions. Isolated structures as well as series of two structures were tested. Among others, the equilibrium scour morphology due to worked wood piles resulted to be considerably influenced by inflow conditions, tailwater level and number of structures in case of series arrangement. Namely, the scour and dune features in the stilling basin of the structure are more enhanced for higher discharge and tailwater and in case of two worked wood piles in series. Moreover, the development of scour and dune geometry was recorded at regular intervals of time for the entire test duration which revealed that the rate of increase of scour area in case of two worked wood piles in*

series is higher in specific intervals during the test depending on the hydraulic conditions. Overall, the results offer important observations and a preliminary analysis of the local scour phenomena around worked wood piles, aiming at providing a basis for future design guidelines.

Keywords: *river restoration structures, scour evolution, equilibrium morphology, inflow conditions.*

Paper ID: 1499

Calculating Local Pier Scour in Coarse-bed Streams

D.C. Froehlich[1]
[1]*Consulting Engineer, Cary, North Carolina, USA*
E-mail: dcfroehlich@aol.com

Abstract: *Streams can be divided into two broad classes based on the predominant composition of the alluvial material in which they are formed: sand-bed and coarse-bed. Coarse-bed streams are formed primarily in gravel and larger sediments. They are common in mountain and foothill regions, mainly where igneous and metamorphic bedrock exists, and in areas of glacial-fluvial deposits. A feature that characterizes nearly all coarse-bed streams is a surface covered by large sediment particles compared to those forming the substratum –an armor layer. The presence of an armor layer has a considerable influence on the physical processes that scour material from around a bridge pier. Several methods exist for estimating local scour depth at bridge piers. Data collected in small-scale laboratory experiments in sand-filled flumes, which hardly resemble conditions in coarse-bed streams, are the basis for most. Fortunately, recent on-site measurements of local pier scour in coarse-bed channels supply a collective data set used in this analysis to develop a scour prediction relation for application in streams where armor layers are present. Nonlinear quantile regression provides prediction models for quantile levels of 0.5 (which gives the median value), 0.95, and 0.99. Estimates for the larger quantiles (corresponding to the 95 and 99 percentiles) offer statistically sound safety margins needed for a reliable bridge design that is not overly excessive. The predictive ability of this study's local pier-scour relations for coarse-bed streams provides significant improvements over other commonly used empirical equations.*

Keywords: *Bridge, pier, scour, coarse-bed, empirical model, quantile regression, safety margin*

Paper ID: 1439

Plunge Pool Rock Scour Assessment by Fluid-Solid Coupling

E.F.R. Bollaert[1] & M. Grünzner[2]
[1]*AquaVision Engineering Llc, Ecublens, Switzerland*
[2]*Flow Science Deutschland GmbH, Rottenburg, Germany*
E-mail: erik.bollaert@aquavision-eng.ch

Abstract: *This paper presents a novel approach for detailed 2D numerical assessment of rock scour in plunge pools or unlined channels downstream of hydraulic structures. The approach is based on an automated coupling between the 3D hydraulics and the turbulence of an impacting jet (or any other turbulent flow environment) and the detailed bathymetric evolution with time of the rocky bottom that is being scoured, for a given flood event. Detailed flow hydraulics are thereby computed by the FLOW-3D® CFD software, allowing sound implementation of air entrainment and defining the main flow parameters at the water-rock interface. Bathymetric evolution with time of the eroding rock mass downstream of the hydraulic structure is computed by the rocsc@r® software, a cloud-based digital tool implementing the latest and most widely used computational methods for scour prediction. This novel fluid-solid interactive approach has been tested and applied to large-scale laboratory experiments of scour hole formation in broken-up rock, generated by rectangular-shaped oblique jets*

impinging in a plunge pool made of cobblestones, performed at the laboratory of the Civil Engineering Department at Stellenbosch University, South Africa (Bosman and Basson (2020), Bosman (2021)). The scour formation observed during the experiments has been reproduced by the FLOW-3D® - rocsc@r® numerical coupling. Benefitting from a dedicated, user customizable and fully automated interface between both software programs, this coupling requests only very short computational times and constitutes a cutting-edge tool available to engineers for real-life applications and projects. The results obtained for different computational methods of scour formation and a set of CFD modelling parameters are pointed out.

Keywords: *fluid-solid coupling, novel software, plunge pool, rock scour*

Paper ID: 1441

Bank scour protection using spur dyke in a meandering channel under low flow velocity

Nishank Agrawal[1] & Ellora Padhi[1]
[1]*Shiv Nadar University, Greater Noida, Uttar Pradesh, India*
E-mail: na886@snu.edu.in

Abstract: *In geophysical flows, the channel often starts to meander when it encounters a plan terrain. Owing to higher flow velocity, it tends to erode the channel boundary (bed and banks) that comes across its path. Scouring in a meandering channel primarily depends on channel boundary, flow condition, and sediment properties. To date, numerous works have been conducted to understand the velocity distribution and scouring pattern in a meandering channel under varying discharge conditions. However, only a handful of work has been reported on the scour pattern under low discharge conditions in a meandering channel. Therefore, in this paper, an experimental study was conducted under a very low discharge in an 80° (central angle θ) bend to analyse the scour pattern. Investigation of the scour profile reveals that maximum scour occurs at the downstream side of the bend apex rather than the bend apex. Hence, an attempt has been made to counter the scour using the spur dyke. In doing so, a non-submerged spur dyke was installed at three different locations θ = 0°, 40°, and 60° of bends with three orientations α = 60°, 90°, and 120° for approaching and developed flow conditions to identify the best location and orientation of the spur dyke. Observation suggests that spur dyke at θ = 60° with α = 90° and spur dyke at 'θ' = 40° with 'α' = 60° provide the best results for approaching and developed flow conditions, respectively, in terms of preventing scour under low flow velocity.*

Keywords: *Scouring, Meandering channel, Spur dyke, Dyke orientation, Low flow velocity.*

Paper ID: 1477

Numerical Modeling for Optimization of the Aspect Ratio of Submerged Vanes for the Purpose of Sediment Deflection in Rivers

V. Chauhan[1], R. Chavan[2] & G.D. Singhal[1]
[1]*Shiv Nadar University, Greater Noida, India*
[2]*Maulana Azad National Institute of Technology, Bhopal, India*
E-mail: vc651@snu.edu.in

Abstract: *In order to create a secondary circulation (spiral flow) in the downstream direction to their trailing edge, submerged vanes are tiny aspect ratio (H/L) devices that are positioned vertically on riverbeds. Since this secondary circulation is driven by the vane-induced tip vortex, submerged vanes are best recognized for changing the alluvial river-bed profile. Finding the right aspect ratio as a function of the angle of attack with the approach flow is one of the current issues with using the submerged vane technique. In this context, the current study aims to investigate the best vane length to*

be used, based on a specific angle of attack with the flow. A pair of rectangular vanes with a set height is among the studied vanes. Only 15^0 and 40^0 angles will be examined in the current study because these are the ideal minimum and maximum recommendations made by earlier studies. The vanes' length to height ratios that were taken into consideration for this investigation were 2, 3, and 4. The purpose of the study is to determine how these variables affect the strength of the secondary circulation and sediment deflection downstream of the vanes. The flow field around and past the submerged vane is investigated using the Flow-3D Hydro numerical model. The most ideal aspect ratios for a low Froude number of 0.168 are 0.25 and 0.33 at 40 and 15 degrees of attack, respectively. The most ideal aspect ratios with a high Froude number of 0.303 are 0.33 and 0.25 at 40 and 15 degrees of attack, respectively.

Keywords: *morphology, inland navigation, scouring, secondary circulation, sediment deflection, vorticity.*

Paper ID: 1475

Experimental Evaluation of Drag Force on Different Shapes of Pontoons at Different Water Stream Velocities at Specified Loads and Submergence

Ms. R. B. Deogade[1], Smt. H. R. Khandagale[2], Shri. M. Someshwara[3]
[1, 2, 3]Central Water and Power Research Station, Pune, India
[1]deogade_rb@cwprs.gov.in, [2]harshali.rk@cwprs.gov.in, [3]milan.s@cwprs.gov.in

Abstract: *The ability to cross both natural and man-made obstacles have always been crucial for good mobility and is vitally important for various operations. The series of Pontoons are used as floating bridge to allow the transport vehicles and personnel over water and marshy grounds. Floating bridges are cost-effective solutions for crossing large bodies of water with unusual depth. Friction drag is observed when relative motion exists between pontoon and fluid. In view of this drag force is required to be evaluated correctly so that arrangement to fix the pontoon can be accurately designed considering the load and safety factors. For this required to carry out experiment to measure drag forces acting on the pontoons due to water resistance at various weights and water velocities. To study and collect numerous data, three pontoons of various shapes and materials were chosen for experiments. The drag force of pontoons is measured at 0, 5 and 10 degrees by add on various loads and submergence levels. During the experiments, a pontoon freely suspended to Rating Trolley when towed at a predetermine speed in straight open water tank, the drag force induced is measured by precise load cell. Experimental outcomes of all three pontoons for drag forces at 0, 5 and 10 degrees at various submergences and velocities will limit its application in specific velocity range and weight applied on it. Experimental analysis indicates the drag force is directly proportional to the frontal resistive area, water velocity and depth of submergence.*

Keywords: *Drag force, Pontoon, Submergence, Rating Trolley.*

Paper ID: 1476

Performance Testing of Acoustic Doppler Current Profiler Used For Stream Flow Measurement

Ms. R.B. Deogade[1], Smt. H. R. Khandagale[2] & Shri. M. Someshwara[3]
[1, 2, 3]Central Water & Power Research Station, Pune, India
[1]deogade_rb@cwprs.gov.in, [2]harshali.rk@cwprs.gov.in, [3]milan.s@cwprs.gov.in

Abstract: *As water demand is steadily increasing worldwide, it is very important to measure the discharge accurately for sustainable water resources management such as flood control, water*

distribution, hydraulic structures design, and hydro-environment management. Since decades the cup and propeller type current meters are used for the measurement of stream velocity, but due to emergent technology, Electromagnetic, Acoustic Doppler Current Profiler (ADCP), etc. are used for measuring the stream flows. These instruments have higher edge in accuracy, consistency, and reliability. Quantitative information on performance and validation in the field of discharge measurement is rare. Hence performance testing of such instrument is essential prior to conducting measurements in fields. The objective of the study is to ascertain the accuracy in velocity measurement of these instruments with the standards. The ADCP transmits a constant high frequency sound wave into the water that ricochets off particles suspended in the moving water and reflect back to the instrument with a slightly lowered frequency due to the Doppler Effect. The performance testing of these sophisticated instruments is done in the straight open channel by Current Meter Rating Trolley. The ADCP deployed on the trolley will be towed at a predefined speed of the Rating Trolley and the corresponding velocity indicated by the sensor will establish the relation. This indicates the performance of the ADCP and its appropriate working in terms of accuracy.

Keywords: *Acoustic Doppler Current Profiler, Current Meter Rating Trolley, Doppler Effect, Water Velocity.*

Paper ID: 1447

Porous Concrete Slabs Underlain by Polypropylene Fabric for Restoration of Beaches

N.N. Pillai[1], M.L. Kansal[2], & R. Ramkrishnan[3],
[1]*Emeritus, School of Engineering, Amrita University, Coimbatore, India*
[2]*Water Resources Development and Management, I.I.T. Roorkee, Roorkee, 247667, India*
[3]*Scientist C, Directorate of Forest Education, Burnihat, 793101, Assam, India*
Corresponding E-mail: mlk@wr.iitr.ac.in

Abstract *Beaches in India and elsewhere are getting destroyed at an alarming rate as many small ports are being built and sea walls are coming up on the banks. There is an urgent need to stop it. In 1966, Posey used cylinders of net filled with rock (called rock sausages) underlain by a reverse gravel filter for safety against scour in hydraulic structures. Pillai used a bed of stones enclosed in nets underlain by a reverse filter to allow overflow over a sand embankment. Later it was understood that a Polypropylene Fabric (P.F.) could be effective as a filter, and a bed of polypropylene bag filled with sand (like geo-tubes) can be used to prevent scour. However, the P.F. get weathered when exposed to sunlight. A breakthrough came when porous concrete blocks with high porosity, high strength and seawater resistance were developed and were used underlined by a P.F. An attempt was made in an experimental study on the Cheriathura Beach at Trivandrum. Porous concrete blocks 0.5 m X 0.5m, 10 cm thick were cast and arranged at a time of low tide, covering an area of 10 m x 5 m (with the P.F. underneath). When the waves acted frontally during high tide, this rode smoothly on the pavement, had their run-up, and later receded without displacing the slabs. This showed that porous concrete slabs could be a good armour for protecting the beach. The side slabs got displaced with lateral waves because no sidewall could be provided in the narrow strip of protection. It is felt that the porous concrete slabs can be placed deep in the beach so that during the erosive phase, the slab could function as armour, and when there is accretion, there would be deposition of sand on the slabs maintaining the beach.*

Keywords*: Beach restoration, Porous Concrete, Polypropylene fabric.*

Paper ID: 1501

Predicting Density-driven Exchange Flows through the West Crack Breach of the Great Salt Lake Causeway with CFD and ANN

B.M. Crookston[1], M. Rasmussen[1], E. Larsen[2], H. Kartchner[2] & S. Dutta[2]
[1]*Department Civil and Environmental Engineering, Utah Water Research Laboratory, Utah State University, Logan, Utah, USA*
[2]*Department Mechanical and Aerospace Engineering, Utah State University, Logan, Utah, USA*
E-mail:brian.crookston@usu.edu

Abstract: *The Great Salt Lake is of environmental and economic value yet is threatened by various factors including drought and water diversion for irrigation purposes. Management efforts are required to preserve this saline lake; such efforts include accurately estimating the exchange flow through an opening in a railroad causeway that divides the lake. This study investigated two modeling approaches for predicting these discharges, a physics-based computational fluid dynamics model and a data-driven artificial neural network model. Good agreement was found between both models, and the advantages each provides to water management efforts are noted. Results indicate that, regardless of the modeling tool, accurate field data is invaluable when studying a hydraulic structure.*

Keywords: *Exchange Flow, CFD, ANN, Great Salt Lake.*

Paper ID: 1433

Hydrodynamics of Permeable Horseshoe Obstacle in Fluvial Environment: a Physical Modelling

H. Chanson[1], H.K. Lee[1] & W. Johnson[1]
[1]*The University of Queensland, School of Civil Engineering, Brisbane, Australia*
E-mail: h.chanson@uq.edu.au

Abstract: *Along Australian inland waterways, a number of man-made horseshoe obstacles were built for centuries. The permanent structures interacted with the streamflow across a wide range of water discharges including when they are fully-submerged. Their use ranged from water holes and fish trap at low to moderate flows, to large bed roughness and turbulent manipulation at large water discharges. The aim of the study was to gain a sound physical understanding of the hydraulic operation of horseshoe obstacle across a broad range of submergence ratio, ranging from low flows and emergent obstacle to major floods with fully-submerged structures. In this laboratory study, the physical modelling of permeable horseshoe obstacle was undertaken under controlled flow conditions, based upon a Froude similitude. Two physical models were built corresponding to a 5:1 and 20:1 geometric scaling ratio for typical riverine structures. The scale models were 3D-printed with random pattern and a porosity of 0.27, close to the porosity of rockfill material. The physical observations included visual observations, three-dimensional free-surface profiles and detailed velocity measurements. The physical observations showed a broad range of flow patterns, depending upon the submergence ratio. The porosity of the obstacle facilitated some interactions between the seepage and recirculation region, leading to changes in the wake region and its turbulence, compared to a impervious obstacle with the same shape.*

Keywords: *Physical modelling; Horseshoe obstacle; Wake region; Riverine interactions.*

Paper ID: 1434

Near-full-scale physical modelling and open-channel flow velocity in a fish-friendly culvert with full-height sidewall baffles

J. Hu[1], Y. Li[1] & H. Chanson[1]
[1]*The University of Queensland, School of Civil Engineering, Brisbane, Australia*
E-mail: youkai.li@uq.edu.au

Abstract: *A near-full-scale physical modelling was performed on the oscillation and instability of open-channel flow in a fish-friendly culvert equipped with full-height sidewall baffles. High-resolution measurements of the instantaneous flow velocity were conducted using a Vectrino+ acoustic Doppler velocimeter (ADV). The physical results were marked by the existence of some low-frequency seiche phenomenon. A well-established triple decomposition technique was applied to the time series of free-surface and velocity time-series. The low-pass components confirmed a unique flow structure, consisting of a high-velocity zone in the main channel and a low-velocity flow reversal within the lateral cavities. The band-pass components corresponded to the low-frequency flow oscillations, highlighting the complicated transverse interactions between the lateral cavity and the main channel. The high-pass velocity components were related to the 'true' turbulence characteristics. The current study provides some further insights into the sustainable design of culverts to assist with upstream fish migration in man-made and natural fast waterways.*

Keywords: *Physical modelling; Sidewall baffle; Fish-friendly culvert; Hydrodynamics; Triple decomposition.*

Paper ID: 1484

Physical and Numerical Model Studies of Hirakud Dam Additional Spillway –A case study

V. P. Gadhe[1] & S. R. Patnaik[1]

[1]*Central Water and Power Research Station, Pune -411024, India*
E-mail: *vpgadhe@gmail.com, sangeeta_patnaik@rediffmail.com*

Abstract: *Stilling basins are frequently used as energy dissipaters to protect the downstream hydraulic structures from scouring. The stilling basin uses the hydraulic jump as the hydraulic element for dissipating energy in a specific structure. A Stilling basin type of energy dissipator is mainly used for economic reasons; therefore, the Stilling Basin II type has been used on high and earth dam spillways and large canal structures. Basin II type design aims to reduce the hydraulic jump length by installing accessories, such as chute blocks and dentated sill in the stilling basin.*

Hirakud Dam is one of the multipurpose projects. In order to safely pass the reassessed additional flood of Hirakud dam, two additional spillways at suitable locations were proposed. One of these was proposed on the main spillway's left bank (Phase-I). Hydraulic model studies of this left bank additional spillway was carried out at CWPRS. This paper is focused on the physical and numerical model studies to assess the performance of the proposed BASIN II type energy dissipator and to finalize the design of the additional spillway on the left bank (Phase-I). Physical model study results revealed that the performance of the original spillway design with energy dissipators was unsatisfactory as the length of the stilling basin provided was insufficient to contain the hydraulic jump. Hence the modification in the length of stilling basin was suggested by the CWPRS. The physical model studies have also been carried out on the modified design and finalized the design of spillway and energy dissipator. The numerical model studies validate the simulation results with the physical model study results for the original and

modified design. Numerical Model studies have been carried out on the licence copy of Flow 3D CFD software at CWPRS.

Keywords: *Additional Spillway, Chute block, Dentated sill, Flow 3D CFD, Type II Stilling Basin,*

Paper ID: 1449

Flow Over Orifice Spillway: Physical and Numerical Study for Spillway Profile Design of Hydroelectric Project

Sushma Vyas[1] & Y. N. Srivastava[2]
[1]*Scientist C, Central water and power research station, Pune, India*
[2]*Additional Director, Central water and power research station, Pune, India*
E-mail: vyassush13@gmail.com

Abstract: *Many high dams on the upper Himalayan rivers which have very steep bed slopes are constructed with orifice type of Spillways with low level sluices for flood disposal and flushing of sediments. To assess the performance of spillway physical model studies were conducted on a 1:50 scale 2-D sectional model. Physical model studies can have a wide range of outcomes depending on the purposes of the model study, although many physical models frequently analyse properties such as velocity patterns, discharge rating curves, water surface profiles, and pressures. The physical measurements highlighted that the spillway is subjected to negative pressures. To overcome this, different spillway profile options were verified using numerical methods. For this, a commercially available computational fluid dynamics (CFD) program FLOW 3D was used which solves the Reynolds-averaged Navier-Stokes equations along with RNG turbulence closure model. Numerical modelling is quite beneficial when used by the designer in combination with the physical modelling. Numerical modelling enables designers to evaluate several possibilities until they find the hydraulically optimal solution, which can then be evaluated on a physical model, reducing the expense of physical model studies. In order to validate the numerical modelling results, the measured data from physical model studies for the original profile of spillway is used. Discharge passing through spillways, water surface profiles and pressures were used to compare the results of the physical model and the numerical model. When it was seen that there is reasonably good agreement between the physical and numerical model results, then all other alternatives were analyzed numerically and an alternative having a hydraulically optimal solution was selected to study on the physical model.*

Keywords: *CFD, Flow 3D, Numerical modelling, Physical modelling, Spillway*

Paper ID: 1470

Laws of turbulence and the estimation of turbulent kinetic energy budget for flow through a degraded channel-bed

Mosedul Sarkar[1]& Sankar Sarkar[2]
[1]*Indian Statistical Institute, Kolkata, India 700108*
[2] *Indian Statistical Institute, Kolkata, India 700108*
E-mail: mosedulsarkar2014@gmail.com

Abstract: *This present study focuses on the laws of turbulence and estimation of the turbulent kinetic energy (TKE) budget for flow over a bimodal degraded bed under equilibrium conditions. The analyses of turbulence were performed based on the three-dimensional velocity data captured by using a Vectrino velocimeter. The streamwise velocity and Reynolds shear stress profiles show an excellent agreement with the existing literature. In view of the advancement in the direction of bed degradation, the measured velocity data captured at different streamwise locations (before, on and after the maximum equilibrium depth of degradation) were further processed to explore the scaling laws of*

turbulence and TKE budget based on structure functions data. To this end, the second-, third-, and mixed third-order velocity structure functions were estimated. The structure functions data indicated the existence of inertial subrange within lower ranges of the time-lags and confirmed their agreement with the Kolmororov's 2/3-, and 4/5-, and Monin–Yaglom's 4/3-laws. However, the TKE dissipation rates estimated by Kolmororov's 4/5- and Monin–Yaglom's 4/3-laws of turbulence show significant differences in their values. The TKE budget results show that in the vicinity of the degraded bed, TKE dissipation rate is more than that of TKE production and turbulent diffusion rates. On the other hand, the pressure diffusion has larger negative values in the near-bed. However, all the turbulence parameters show no significant differences at as one moves from the initial bed-level (bed-level measured before the detachment started) to the free-surface.

Keywords: *Degraded bed, Open channel flow, structure functions, Scaling laws, turbulent flow, TKE budget*

Paper ID: 1464

Numerical modelling of scour and flow field for two different arrangements of piers using SSIIM model

Sushant Kumar Biswal1 & Animesh Das1
[1]*National Institute of Technology Agartala, Tripura, India*
E-mail: sushantb69@gmail.com
E-mail: animesh.nit14@gmail.com

Abstract*: In this work, a numerical model Sediment Simulation In Intakes with Multiblock option (SSIIM) is developed to analyse the effect of pier spacing on the scour process around side by side and staggered piers. Both* $\kappa - \varepsilon$ *and* $\kappa - \omega$ *turbulence models are employed to solve the eddy viscosity flow equations and their performances are evaluated. From the initial observations it is concluded that, when the spacing between the piers is negligible, then both the piers behave like a single pier and the scour depth is found to be more than 42% than that of a single pier. However, when the clear spacing between piers is more than 1.5 times pier diameter* (D) *for both arrangements the piers behaved independently. Furthermore, the numerical model is skill-assessed for two different piles spacing* (S/D) *in side by side piers and in staggered arrangements. While the model is effective in reproducing the scour depth in front of piers for side by side piers, it overestimated the bed scour depth between the piers, however, it underestimated the scour depth in front piers in the staggered arrangements. The results show that the SSIIM program can be used efficiently for simulating the scouring in natural rivers and good agreements are achieved between the piers in different arrangements.*

Keywords *: Circular pier, Flow dynamics, Numerical method, Scour, SSIIM2.0*

Paper ID: 1493

Scour Geometry and Dune Formation Characteristics around Customized Structure in Channels

Rajib Das[1], Subhasish Das[1], Michele Palermo[2], Deep Roy[3] & Simone Pagliara[4]
[1]*School of Water Resources Engineering, Jadavpur University, Kolkata, India*
[2]*Department of Energy, Systems, Territory and Construction Engineering, University of Pisa, Pisa, Italy*
[3]*Department of Civil Engineering, National Institute of Technology Tiruchirappally, Tiruchirappally, India*
[4]*Laboratory of Hydraulics, Hydrology and Glaciology (VAW), ETH Zurich, Zurich, Switzerland*
E-mail: rajib.das@jadavpuruniversity.in

Abstract: *In this study, an attempt is made to verify the scour geometry and location of the dunes at a channel downstream where various structures are used as flow barriers. Various structures such as a*

square structure with a vane attached to one side, a rhombus and two similar structures positioning eccentrically, are used to examine scour geometry and dune formation. Overall, six Clearwater tests are performed, three of which are of single structures and three are a combination of two similar structures. All tests are performed keeping approach flow depth, discharge and bed material fixed. The main focus of the study is to assess the scour on both sides of such structures, relocating sand downstream and their volume. The scour depth, length, planar area and volume are measured and calculated. It is observed that the scour volume and relocating of sediments are significantly more around double structure arrangement. Also in double structure tests, sediment relocation is much greater on the eccentric structure side. As a result, on the eccentric structure side, the scour planner area and the angle of sediment relocation are significantly higher compared to the non-eccentric side. To examine further, the scour hole characteristics and the volumes of sediment relocation are computed from observed data.

Keywords: *eccentricity; scour depth; sediment relocation; square structure.*

Paper ID: 1491

Estimation of Maximum Scour Depth and Scour Pattern Around Submerged Spur-dike

Manish Pandey[1] and Manali Pal[2]
[1,2]*Department of Civil Engineering, National Institute of Technology Warangal, Warangal, India*
E-mail: *manishpandey3aug@gmail.com, manalipal@nitw.ac.in*

Abstract: *Scouring due to turbulent flows is a process that naturally happens around river structures, including river training or riverbank protection works such as spur dikes, submerged vanes, etc. present experimental study aims to estimate the maximum scour depth around submerged spur dikes, which have not been studied as extensively. In this study, we checked the influence of different flow conditions, spur dike dimensions and submergence conations on maximum scour depth at equilibrium scour conditions. The dependence of the criteria for scour initiation and scour patterns on the dike dimensions, such as the dike length and height, has been assessed. We also checked the previously proposed equations for maximum scour depth at submerged spur dike using present and previous (collected from literature) experimental data. The dependency of maximum scour depth on some hydraulic and geometric control parameters is also discussed. In addition, a new equation is also proposed to calculate the maximum scour depth around submerged spur dike at equilibrium condition, which shows better agreements with present and previous experimental datasets than other equations.*

Keywords: *Clear-water condition, equilibrium scour condition, maximum scour depth, submerged spur dike.*

Paper ID: 1467

An Experimental Study on Scour at Zero Degree Confluent Channels

Mohd Faisal Ansari[1], Mohammad Aamir[2] & Zulfequar Ahmad[1]
[1]*Indian Institute of Technology Roorkee, Roorkee, India*
[2]*Chaitanya Bharathi Institute of Technology, Hyderabad, India*
Email: fai.ansari1991@gmail.com

Abstract*: This paper presents an experimental study on scour depth at the confluence of a mobile and a rigid bed open channel. Substantial momentum difference between the confluent channels controls flow hydrodynamics and mixing in the confluence hydrodynamic zone. Scour at the confluence of the channels is a threat to the stability of the structure separating the two channels. Results for bed*

morphology and maximum scour depth at the confluence for different hydraulic and geometric conditions in the channels are presented. Data analysis shows that the discharge ratio (ratio of discharge in mobile bed and total discharge downstream) and tailwater conditions are key factors affecting the bed morphology and maximum scour depth. The maximum scour depth decreases by increasing the discharge ratio and tailwater depth.

Keywords: *bed morphology, confluence, hydrodynamics, open channel, scour*

Paper ID: 1444

Experimental optimization of a gravel trap in a gallery

S. Erpicum[1], N. Claude[2], B. Dewals[1], P. Archambeau[1] & M. Pirotton[1]
[1]*Liege University – Hydraulics in Environmental and Civil Engineering Research Group, Liege, Belgium*
[2]*Electricité de France – Hydro Engineering Center, La Motte Servolex, France*
E-mail: s.erpicum@uliege.be

Abstract: *A partly unlined pressurized gallery supplies water to a hydropower plant. Since rocks and gravel may detach from the gallery walls, two gravel traps have been built on the gallery invert, downstream of the unlined sections. During gallery and powerplant inspections, it appears that the gravel traps are empty while gravel is found in the tail channel downstream of the turbines. Clear evidence of turbine damaging by solids impact is also observed. To understand why the gravel traps are ineffective and to define solutions to be implemented on site to increase their trapping efficiency, a 1:7.7 scale physical model of the gravel traps has been built and operated. The tests clearly demonstrated that the short-term trapping efficiency of the gravel traps is good since 98% of the gravel arriving upstream falls into the traps. However, the gravel traps are ineffective in keeping the gravel over long periods. This is due to two effects. In pressurized conditions, helicoidal vertical currents developing at the downstream extremities of the trap and turbulent velocity fluctuations extract the gravel from the trap. In addition, during gallery dewatering (for maintenance and inspection), temporary free surface flow can transport gravel downstream of the trap. Non-structural solutions to the problem have been identified, optimized and validated on the scale model. They consist in creating above the traps a pseudo-bottom aligned with the gallery invert, using horizontal bars placed transversally to the flow, and in compartmenting the traps with vertical walls. The paper presents in detail the physical model and all the tests and measurements performed to quantify the problem and assess the efficiency of the modification solutions.*

Keywords: *Hydropower, Physical modeling, Rock trap, Scale modeling, Trapping efficiency, Unlined tunnel.*

Paper ID: 1438

Investigation on near bed flow features over a water-worked gravel bed

Ellora Padhi[1] & Nishank Agrawal[1]
[1]*Shiv Nadar University, Greater Noida, Uttar Pradesh, India*
E-mail: ellora.padhi@snu.edu.in

Abstract: *One of the key problems in the research on fluvial hydraulics is to correctly reproduce the flow dynamics over a riverbed in laboratory experiments. Generally, while performing experiments, attention is paid to specific hydraulic conditions related to the flow regime, sediment transport, and bed erosion. In this context, the sediment size of the riverbed to be analysed plays an important role. It is a common practice to create the bed manually using a screeded mixture of sediments with random orientation that could be very different from that observed in natural riverbeds. To overcome this issue, a manually created gravel bed is required to be water worked before performing the experiments to get*

a bed surface, which resembles to that of a natural gravel bed river. Such a bed is termed as water-worked gravel bed (WGB). In recent past, even though the bed topography of WGB has been intensively studied, its effects on the flow features, which govern the near-bed flow field, are still to be explored under varying flow rates. In this study, the double averaged (DA) turbulence characteristics such as velocity, Reynolds shear, and normal stresses, and form induced shear and normal stresses, are analysed for three different flow rates. A comparative study infers that with an increase in flow rate the magnitude of all flow features increases, especially in the near-bed flow zone. The reason for such occurrence is attributed to a higher flow rate that enhances the near-bed turbulence, and in turn the velocity fluctuations.

Keywords: *Double-averaging methodology, Turbulence characteristics, Water-worked gravel bed.*

Paper ID: 1436

Reynolds Stress Modelling of Supercritical Flow in a Narrow Channel

S. Kadia[1], N. Rüther[2] & E. Pummer[1]
[1]*Department of Civil and Environmental Engineering, Norwegian University of Science and Technology, 7491 Trondheim, Norway*
[2]*Chair of Hydraulic Engineering, Technical University of Munich, Arcisstr. 21 80333 Munich, Germany*
E-mail: subhojit.kadia@ntnu.no; subhojitkadia@gmail.com

Abstract: *One of the most effective techniques to combat reservoir sedimentation, especially for small to medium-sized reservoirs, is the installation of Sediment Bypass Tunnels (SBTs). SBTs are designed for supercritical narrow open channel flow conditions. In such narrow channels, the walls and the free surface boundaries can influence the flow characteristics and form secondary currents. These vortex structures can alter the longitudinal velocity distribution and form velocity dip. Besides, they can also alter the bed shear stress distribution and influence the sediment transport. Such complex phenomenon can be modelled using Reynolds Stress Models (RSMs). In this study, the full Launder Reece and Rodi (LRR) pressure-strain model was implemented in OpenFOAM® and the simulated results were compared with former experimental results for flow depth = 0.16 m, Reynolds number = 5.7×10^5, Froude number = 1.84, and channel aspect ratio = 1.25. In addition, the dissipation rate of turbulent kinetic energy at the free surface was modelled following literature. The simpleFoam solver, which is based on the SIMPLE algorithm, was used for the steady state simulation. The predicted velocity distributions, secondary currents, velocity dip, turbulence intensity distributions, and lateral variation in bed shear stress are consistent with literature. The results show that the simulated maximum longitudinal velocity and the cross-sectional average bed shear stress are 0.34% and 0.74% lower than the observed values. An intermediate vortex was developing between the free surface vortex and the bottom vortex for the tested case. However, it remained as a part of the free surface vortex.*

Keywords: *RSM, secondary currents, sediment bypass tunnel, supercritical flow, velocity dip.*

Paper ID: 1453

Importance of Risk and Hazard Assessment of River Projects in India

Jitesh N. Vyas, Supriya Nath and R. B. Deogade
Central Water and Power Research Station, Pune-411024
E-mail: jiteshvyas33@gmail.com

Abstract: *Rivers are the lifeline for every country as they provide water for proliferation, sustenance and propagation of life since time immemorial. With increasing population pressure, water demand increased for drinking, irrigation, and industrial consumption. This water deficit required utilization of available water resources in the best possible manner through various hydrological projects like construction of dams, weirs, barrage, etc. There is a disparity in water availability between north and*

south India as most of the perennial rivers are situated in the north while seasonal rivers in south India. To resolve this disproportionate distribution and to maintain the balance of water between flood and drought-prone rivers, interlinking of rivers is proposed. This proposal has garnered a lot of attention from scientists and environmentalists around the world. These human interventions affect the ecological flow of rivers. There are various risks associated with these projects which require prior assessment. In this study, risks and environmental hazards associated with river projects in India have been discussed with a special focus on the ongoing river projects like the interlinking of rivers. Various factors which determine the feasibility of the proposed projects were studied. There are many risks associated with the change in gradient and environmental flow of rivers with the construction of such projects. We have reviewed risks on humans, like displacement of population and impact on their livelihood as reported for various projects. There are various social implications of such projects which need to be assessed and their mitigation measures need to be undertaken.

Keywords: *environmental flow, gradient, hydrological projects, interlinking of rivers, social implications, water quality*

Paper ID: 1430

Designing Smooth Mixed-Geometry Canal Transition

D.C. Froehlich[1]
[1]*Consulting Engineer, Cary, North Carolina, USA*
E-mail: dcfroehlich@aol.com

Abstract: *When a canal's size or shape changes, usually over a short distance, a section of the channel, known as a transition structure, is needed to connect the waterway's two stretches. Fifth-degree parametric equations are developed to calculate the cross-section dimensions and bed centerline elevations (thus, the geometric surface coordinates) between the two ends of a warped transition structure in a water-supply canal. The parametric modeling approach provides a smooth representation of the mixed geometry that results from terminal sections having vastly different shapes. A generalized cross-section defined by four parameters enables a straightforward model of various forms ranging from trapezoids to semi-circles. This approach significantly simplifies the interpolation of surface coordinates between the terminal points of a transition structure. It also maintains a smoothness that helps avoid undesirable consequences of channel contractions and expansions. An example is presented that applies the parametric modeling approach to design a significant canal transition where the cross-section changes from a standard trapezoidal shape with rounded bottom vertices to a rectangular section in a steeper aqueduct that carries the flow across a broad valley.*

Keywords: *Canal, water supply, transition, parametrically smooth, mixed-geometry*

ACKNOWLEDGEMENTS

I would like to thank the members of the Hydraulic Structure Committee (HSC) of the IAHR for awarding the conduct of the 9th International Symposium on Hydraulic Structures to the Department of Civil Engineering, IIT Roorkee. I would like to express my sincere thanks to the Local Organizing Committee, International Scientific committee (ISC) and the various local committees for their generous support and help. A warm word of appreciation is also due for all the participants and authors for their enthusiasm, dedication and contributing technical papers. I would like to thank all the reviewers for their valuable support and time in reviewing the manuscripts. Special thanks to Prof. Michele Palermo, Chairperson, ISC; Prof. Brian Crookson, Chair of HSC and Prof. Sebastien Erpicum, Past Chair of HSC for their support and help in organising this event and to act as a member of the Editorial Board of review process.

We are also thankful to all the key note speakers who accepted our invitation and shared their ideas and expertise. The organization of the Symposium would not have been possible without the generous and active involvement of our students, laboratory staff and supporting staff since the inception of ISHS 2022. My gratitude is also towards our Former Professors who accepted our request to participate in the symposium.

Financial support provided by the IIT Roorkee under its Programme of Celebration of 175th year is greatly acknowledged. We wish to acknowledge our Knowledge Partners and Sponsors for their moral and financial support. The support of GetSet Events and Conferences and for developing ISHS 2022 website, Virtual Platform, Hall Management, online Registration Platform etc. is also acknowledged.

Sincere thanks are also due to Utah University for providing Utah State University Digital Commons platform for submission and reviewing the abstract and the papers and also for publishing the proceedings.

I also wish to thank all those who were involved and cooperated, either directly or indirectly, to make this Symposium a great success.

Zulfequar Ahmad
Conference Chair

KNOWLEDGE PARTNERS

Central Water Commission

Central Water Commission (CWC) is a premier Technical Organization of India in the field of Water Resources and is presently functioning as an attached office of the Ministry of Water Resources, River Development and Ganga Rejuvenation, Government of India. The Commission is entrusted with the general responsibilities of initiating, coordinating and furthering in consultation of the State Governments concerned, schemes for control, conservation and utilization of water resources throughout the country, for purpose of Flood Control, Irrigation, Navigation, Drinking Water Supply and Water Power Development. It also undertakes the investigations, construction and execution of any such schemes as required.

Indian Society for Hydraulics

Indian Society for Hydraulics (ISH), a technical, educational and non-profiteering national organization, was established in 1992 to serve as a common platform for engineers, scientists and technical personnel working in hydraulics and other allied fields. ISH helps technical personnel stay abreast with the latest development in hydraulics and water resources by organizing regular meets, symposiums, gatherings and seminars. The body also periodically recognizes and awards outstanding achievers in the field.

Central Board of Irrigation and Power

Central Board of Irrigation and Power (CBIP), is a Premier Institution set up by the Government of India in the 1927. CBIP has been rendering dedicated services to the professional organization, engineers and individuals in the country related to Power, Water Resources and Renewable Energy Sectors for the last 90 years. CBIP has grown into an eminent organization of international importance while serving the nation equally with great distinction. The International Commission on Large Dams (ICOLD) is associated with CBIP and a non-governmental International Organization which provides a forum for the exchange of knowledge and experience in dam engineering.

Consulting Engineers Association of India

Consulting Engineers Association of India (CEAI) is the apex body of consulting engineers in India having membership of organisations as well as individuals. The membership represents large, medium and small consultancy companies/ organisations both in the private and public sector and eminent individual consultants. EPC organisations are also members of CEAI since they have specialists in planning, design engineering, apart from construction management. CEAI is the Member Association, of the International Federation of Consulting Engineers, commonly known as FIDIC.

CONSULTING ENGINEERS ASSOCIATION OF INDIA
Creating Value Ethically for Engineers

Cheruthoni dam, India

SPONSORS

NCC Limited

SIGNET™

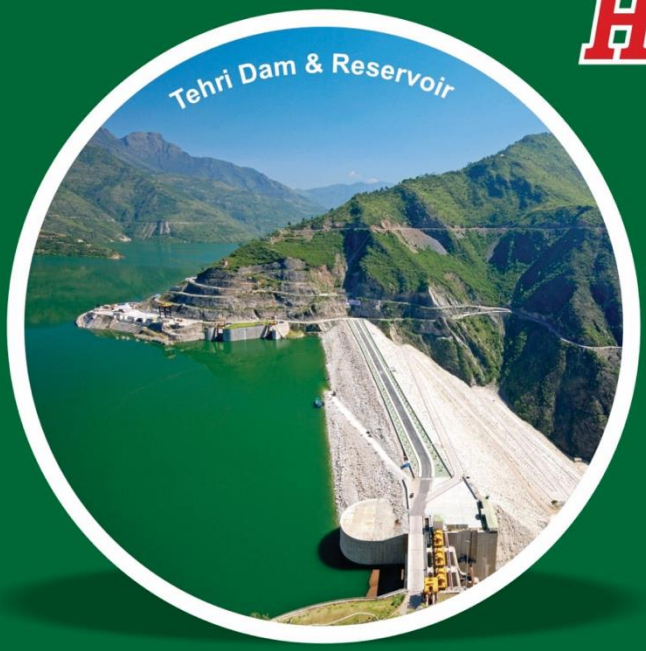
C2S2 Stimulating Sustainability | Cetus Consulting Solutions Services Pvt Ltd
(An ISO: 9001:2008 | ISO 14001:2004 & OHSAS 18007 : 2007 Company)

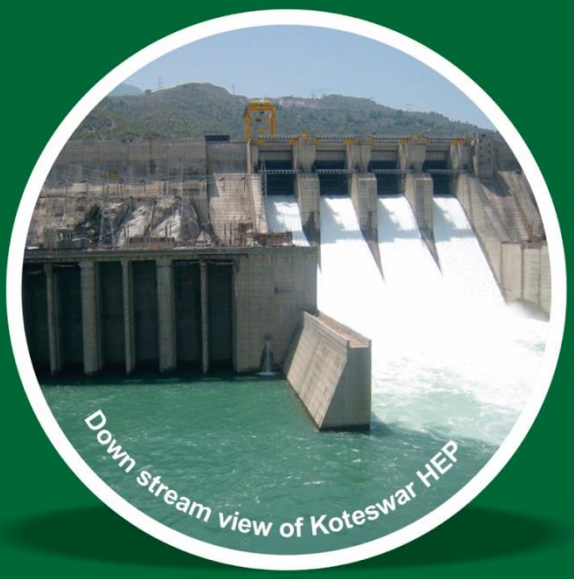
Inland Waterways Authority of India

श्यामा प्रसाद मुखर्जी पोर्ट, कोलकाता
SYAMA PRASAD MOOKERJEE PORT, KOLKATA
Formerly Kolkata Port Trust

टीएचडीसी इंडिया लिमिटेड
THDC INDIA LIMITED

Not just about Hydro Power...





Diversifying with Dedication & Excellence to become a Major Global Player in Power Sector


Solar Power


Thermal Power


Wind Energy


Consultancy Services

टीएचडीसी इंडिया लिमिटेड
THDC INDIA LIMITED
(श्रेणी–क मिनी रत्न, सरकारी उपक्रम)
(Schedule-A Mini Ratna, Government PSU)

गंगा भवन, बाई पास रोड, प्रगति पुरम,
ऋषिकेश–249201 (उत्तराखंड)
Ganga Bhawan, By-Pass Road, Pragatipuram,
Rishikesh-249201 (Uttarakhand)
Website : www.thdc.co.in

Stay Protected from Corona


Wear Your Mask Properly


Regularly wash your hands with soap


Keep Social Distancing

Follow us:- @THDCIL 24×7 @THDCIL_MOP @THDCIL @THDCIL India Limited - Official

www.c2s2.in

Cetus Consulting Solution Services

Nurturing Nature
for Sustainable Infrastructure

Cetus Consulting Solution Services Pvt. Ltd (C2S2) is a professional services firm, dedicated towards finding innovative and appropriate solutions for sustainable development of water and related infrastructure projects. With a committed pool of professional engineers and scientists, C2S2 has a total of about 200 years of impeccable experience portfolio.

Committed to quality delivery system, we at C2S2 have acquired ISO 9001:2008, ISO 14001:2004, OHSAS 180001:2007 certification and adopted the quality norms for all our processes and systems.

Our Area of Expertise:

- **Integrated Water Resources Management**
- **Flood and Disaster Management**
- **Urban and Regional Planning**
- **Port and Maritime**
- **Infrastructure Planning and Management**
- **Environmental Engineering**
- **Social and Community Development Management**

C2S2

Stimulating Sustainability

Cetus Consulting Solution Services Pvt. Ltd.
B-149, First Floor, Chittaranjan Park, New Delhi 110019
Phone: +91 11 41064680, ajay.pradhan@c2s2.in, www.c2s2.in

www.ncclimited.com

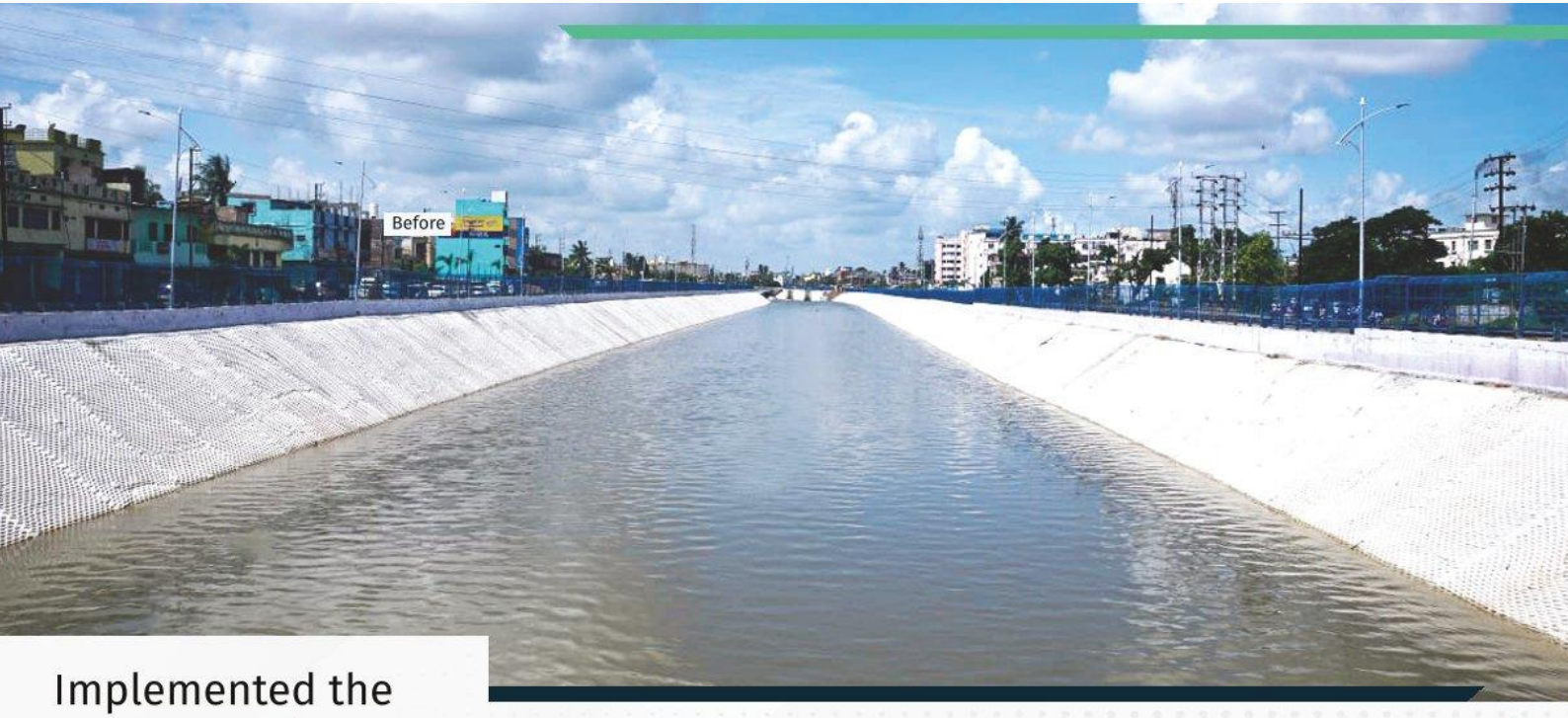
NCC

CROSSING MILESTONES. BUILDING A LEGACY.

NCC has been scripting an amazing success story on a canvas spanning over 4 decades. The single-minded focus on delivering as the nation's partner in progress has seen us build the physical infrastructure that drives economic development; blend experience and expertise to set the benchmark for excellence in execution; and emerge as the second largest listed construction company in India. It's been an epic journey so far, and one that promises to continue well into the future.

BUILDINGS & HOUSING | WATER & ENVIRONMENT | ELECTRICAL | ROADS | IRRIGATION | METALS | MINING | POWER | RAILWAYS

NCC Limited NCC House Madhapur Hyderabad - 500 081 Telangana India T: +91 40 2326 8888



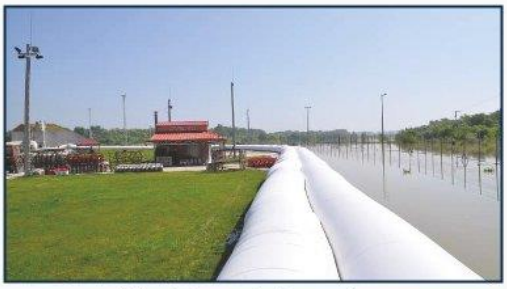
Implemented the

Incomat (Cementitious Concrete Geo-synthetic Mattress)

to transform the century-old Taldanda Canal in less than 3 months saving 70% time over traditional construction methods.

Other Environment & Energy Conservation Solutions by Signet

Cementitious Composite Geo-tex Carpet



No Flood Barrier

Parans Daylight System | Eco Tec Power | Sunpipe | PF

SIGNET™

Signet Enertech Private Limited
8/63, First Floor, CCI Colony, Unnat Nagar No.4,
M.G Road,Goregaon (West), Mumbai - 400 104

022-28771618
022-28772015
88281 28892

info@signet.co.in
admin@signet.co.in

www.signet.co.in

श्यामा प्रसाद मुखर्जी पोर्ट, कोलकाता
SYAMA PRASAD MOOKERJEE PORT, KOLKATA
Formerly Kolkata Port Trust

KOLKATA PORT TRUST 150 years
GLORIOUS PAST · VIBRANT FUTURE

Heads you win Tails you win

It's a win win situation. Anyway you get to win. The all new Kolkata Port gives you the best advantage in the east – in price – in service.

- Primary focus on customer satisfaction
- Competitive pricing to suit customer need
- Widespread rail-road network
- 5th Largest Major Port in terms of tonnage handled
- Single window system
- Extensive storage facilities for diverse cargo
- Modern computerised container terminal, VTMS with AIS and state-of-the-art oil jetties
- India's largest Dry Dock system for ship repair
- Ideal destination for Inland Water Transport
- 50 metres draft available at Sandheads – the highest amongst all major ports in the country & Ideal for Transloading Operations & handling of Cape Vessels

Major incentive for new & additional cargo

श्यामा प्रसाद मुखर्जी पोर्ट, कोलकाता
Syama Prasad Mookerjee Port, Kolkata
पत्तन, पोत परिवहन और जलमार्ग मंत्रालय, भारत सरकार
Ministry of Ports, Shipping and Waterways, Govt. of India
15, STRAND ROAD, KOLKATA-700001
Phone (D): 22306645, 71012214
Website: www.smportkolkata.shipping.gov.in

Glorious Past – Vibrant Future

NATIONAL WATERWAYS

Establishment of Inland Waterways Authority of India

Declaration of 5 National Waterways NW-1 to NW-5

Declaration of 106 new National Waterways under the National Waterways Act, 2016

Declaration of 106 new National Waterways under the National Waterways Act, 2016 along with previously declared 5 National Waterways (NW-1 to NW-5) made it a network of 111NWs across the country.

Benefits of Inland Water Transport

- Low emissions- CO2 equivalent greenhouse gases emission per tonne-km of cargo transportation is 15g by IWT, 28g by Rail and 64g by Road transport.
- Low energy consumption- 1 HP can carry 4000 kg load in water, 150 kg on road & 500 kg on Railways.
- Low fuel cost- 1 liter of fuel can move 24 tonne-km of freight by road, 85 tonne-km by rail and 105 tonne-km by IWT.
- IWT can provide optimal modal mix by intergating river transport with other modes thereby reducing total logistics cost.
- It eases congestion on Road and Rail networks.
- IWT requires very little land acquisition as compared to Road and Rail modes

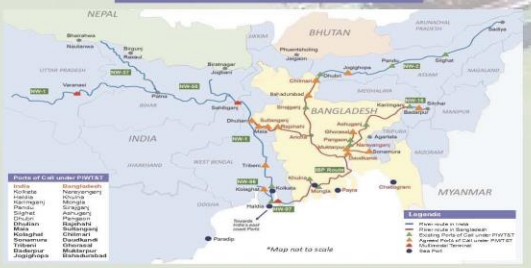
Jal Marg Vikas Project (JMVP)

Capacity augmentation of the 1,390 km of NW-1 from Varanasi to Haldia, is being implemented under the Jal Marg Vikas Project with technical and financial assistance from World Bank (Project cost- Rs. 4633.81 cr.)

Main National Waterways of NER

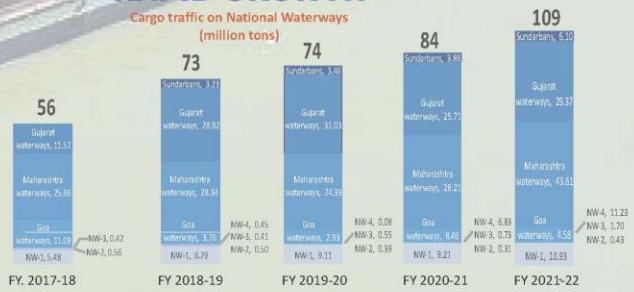
- RIVER BRAHMAPUTRA (NW – 2) (Assam) - 891 km
- BARAK RIVER (NW-16) (Assam) - 121 km
- DHANSIRI / CHATHE RIVER (NW-31) (Assam) - 110 km

IBP ROUTE – A GAME CHANGER



RAPID GROWTH

Cargo traffic on National Waterways (million tons)



Inland Waterways Authority of India

(Ministry of Ports, Shipping & Waterways, Govt. of India)

www.iwai.nic.in

Head Office: A-13, Sector -1, Noida, U.P – 201301, **Tele** : 0120-2544036, 2521684, 2522798, 2521724

Follow us @IWAI_ShipMin facebook YouTube

ISHS 2022

Prof. Zulfequar Ahmad
Conference Chair
Department of Civil Engineering
IIT Roorkee, Roorkee, India-247667
Ph: +91-1332-285423, +91-9012223458
Fax: +91-1332-275568
E-mail: z.ahmad@ce.iitr.ac.in, zulfifce@gmail.com