TEKNIK PENGAIRAN

LAPORAN PENELITIAN MANDIRI

KATEGORI A

MODEL 3-DIMENSI STRUKTUR DAN KARAKTERISTIK TURBULENSI ALIRAN PADA BELOKAN SUNGAI DENGAN DASAR BERGERAK

(3D-modeling of turbulent structure and its characteristics of flow in river bend with movable bed)

oleh:

Dian Sisinggih, ST, MT., Ph.D (NIDN 0019117001)
Ir. Sri Wahyuni, ST, MT, Ph.D., IPM (NIDN 0028055706)
Dr. Ir. Ussy Andawayanti, MS (NIDN 0031016107)
Anggit Gilang Megantara (NIM 185060407111016)

Dilaksanakan atas biaya PNBP Tahun Anggaran 2021 Fakultas Teknik Universitas Brawijaya
berdasarkan kontrak Nomor: 28/UN.10.F07/PN2021
Tanggal 3 Mei 2021

JURUSAN TEKNIK PENGAIRAN - FAKULTAS TEKNIK
UNIVERSITAS BRAWIJAYA
NOPEMBER 2021

HALAMAN PENGESAHAN

Judul Penelitian	:	Model tiga dimensi struktur dan karakteristik turbulensi aliran pada belokan sungai dengan dasar bergerak. *(3D-modeling of turbulent structures and its characteristics of flow in river bend with movable bed)*
Katagori Penelitian	:	**A**
Ketua Peneliti		
a. Nama Lengkap	:	Dian Sisinggih, ST., MT., Ph.D
b. NIDN	:	0019117001
c. Jabatan Fungsional	:	Lektor
d. Program Studi	:	Teknik Pengairan
e. No HP.	:	085727349218
f. Alamat surel (email)	:	singgih@ub.ac.id
Anggota Peneliti 1		
a. Nama Lengkap	:	Ir. Sri Wahyuni, ST, MT, Ph.D, IPM
b. NIDN	:	0028055706
c. Perguruan Tinggi	:	Universitas Brawijaya
Anggota Peneliti 2		
a. Nama Lengkap	:	Dr. Ir. Ussy Andawayanti, MS, IPM
b. NIDN	:	0031016107
c. Perguruan Tinggi	:	Universitas Brawijaya
Lama penelitian keseluruhan	:	6 bulan (seluruhnya)
Biaya penelitian keseluruhan	:	Rp. 15.000.000,- (Lima belas juta rupiah)
Biaya tahun berjalan	:	-

Malang, 10 Nopember 2021

Mengetahui,
Ketua BPP Fakultas Teknik
Universitas Brawijaya

Dr. Ir. Runi Asmaranto, ST, MT., IPM
NIP. 19710830 200012 1 001

Ketua Peneliti

Dian Sisinggih, ST., MT. Ph.D
NIP. 19701119 199512 1 001

Menyetujui,
Dekan Fakultas Teknik

Prof. Ir. Hadi Suyono, ST, MT, Ph.D, IPU, ASEAN ENG.
NIP. 19730520 200801 1 013

IDENTITAS PENELITIAN

1. Judul Usulan : Model tiga dimensi struktur turbulensi aliran pada belokan sungai dengan dasar bergerak. (*3D-modeling of turbulent structures and its characteristics of flow in river bend with movable bed*)
2. Katagori penelitian : A
3. Ketua Tim Pengusul
   (a) Nama Lengkap : Dian Sisinggih, ST, MT., Ph.D
   (b) Bidang Keahlian : Transportasi Sedimen/Hidrolika Sungai
   (c) Jabatan Struktural : -
   (d) Jabatan Fungsional : Lektor
   (e) Fakultas/Jurusan : Fakultas Teknik/Jurusan Teknik Pengairan
   (f) Alamat surat : Jl. Mayjend Haryono 167, Malang 65145
   (g) Telephon / Fax. : (0341) 562454 / (0341) 562454
   (h) E-mail : singgih@ub.ac.id

Anggota Peneliti

a. Dosen

No	Nama dan Gelar Akademik	Bidang Keahlian	Unit Kerja	Alokasi waktu (jam/minggu)
1	Ir. Sri Wahyuni, ST, MT, Ph.D, IPM	Airtanah	Universitas Brawijaya	12 jam/minggu
2	Dr.Ir. Ussy Andawayanti, MS	Konservasi DAS	Universitas Brawijaya	12 jam/minggu

b. Mahasiswa
   Mahasiswa : Anggit Gilang Megantara NIM: 185060407111016

4. Objek penelitian : Ruas belokan sungai/model fisik di laboratorium
5. Masa pelaksanaan penelitian:
   a. Mulai : Mei 2021
   b. Berakhir : Nopember 2021
6. Anggaran yang diusulkan : Rp 15.000.000,- (Terbilang : Lima belas juta rupiah)
7. Lokasi Penelitian : Jawa Timur.
8. Hasil yang ditargetkan :
   - Deskripsi struktur turbulensi aliran dan sirkulasi aliran silang/*secondary flow* serta interaksinya dengan kapasitas angkut sedimen di saluran dan perubahan morfologi dasar saluran.
   - Makalah publikasi untuk seminar/jurnal nasional/internasional
9. Institusi lain yang terlibat : -
10. Keterangan lain yang dianggap perlu :

Penelitian ini merupakan rangkaian dari penelitian jangka panjang sesuai peta jalan (*roadmap*) untuk membangun simulator bencana sedimen yang terpadu untuk mitigasi bencana dan manajemen sedimen waduk (*Integrated Sediment Disaster Simulator for Sediment Disaster Mitigation and Reservoir Sedimentation Management).*

ABSTRAK

Turbulensi adalah properti aliran, tetapi bukan properti fluida sehingga karakteristik aliran turbulen sangat bergantung pada kondisi batasnya. Turbulensi memainkan peran penting dalam aliran dan bertanggung jawab atas penyebaran dan pencampuran panas dan sedimen bahan terlarut atau tersuspensi, polutan, oksigen, dll. Turbulensi ini juga memiliki interaksi yang kuat dengan medan kecepatan rata-rata dan tegangan geser batas. Turbulensi memainkan peran penting dalam pembentukan sel sirkulasi lintas aliran, yang pada gilirannya mempengaruhi distribusi kecepatan dan tegangan geser batas.

Model model hidrodinamika pada saat ini telah memainkan peran penting dalam berbagai disiplin ilmu hidraulika. Dalam bidang hidraulika, model hidrodinamika ini semakin menampakkan eksistensinya yang ditunjukkan dengan semakin seringnya model hidrodinamika dipakai dalam pekerjaan rekayasa teknik. Di sisi akademik, model matematik juga semakin banyak diminati. Sebagian besar topik penelitian yang dilakukan oleh dosen dan topik tugas akhir berkaitan dengan hidrodinamika. Model aliran turbulen telah memberikan kontribusi yang banyak di bidang hidraulik. Metoda penyelesaian masalah turbulensi aliran telah banyak dikembangkan antara lain yaitu metoda *Zero equation model, One equation model, Two equation model, Reynolds stress model* serta masih banyak lagi, namun demikian, karena tingkat akurasi yang diberikan cukup baik, metoda *two equation κ-ε model* lebih banyak dikembangkan.

Pergerakan sedimen di saluran juga berkaitan erat dengan fenomena turbulensi. Terlepas dari pentingnya, sedikit yang diketahui tentang karakteristik turbulensi di belokansaluran terbuka. Fitur aliran yang khas adalah pola *bicellular* dari sirkulasi aliran silang/*secondary flow* dan aktivitas turbulensi di belokan luar yang secara signifikan lebih kecil daripada di saluran lurus. Pada penelitian ini akan dilakukan kajian tentang struktur dan karakteristik turbulensi aliran pada belokan saluran dengan menggunakan *non-linier κ-ε turbulent model.*

Kata Kunci: non-linier κ-ε turbulent model, belokan sungai, angkutan sedimen

RINGKASAN

Dalam mengevaluasi perilaku aliran, seringkali komponen turbulen diabaikan, padahal, pada kenyataannya, aliran air yang terjadi hampir seluruhnya bersifat turbulen. Pengabaian aliran turbulen tersebut seringkali menimbulkan ketidakakurasian yang cukup signifikan dalam menentukan besaran rencana sehingga pada perencanaan bangunan air mempengaruhi stabilitas bangunan air yang bersangkutan, misalnya perlindungan tebing pada belokan sungai, pemasangan krib, gerusan pada pilar jembatan pada sebuah sungai, endsill dari sebuah spillway dll. Kompleksitas fenomena turbulen menyebabkan sulitnya aliran tersebut untuk dapat dihitung dengan menggunakan pendekatan analitik dalam menyelesaikan persamaan aliran. Sementara itu, analisis aliran turbulen secara eksperimen sangat memakan waktu dan biaya yang besar. Oleh karena itu, model numerik menjadi suatu sarana yang lebih banyak dipillih untuk mencapai tujuan tersebut. Namun demikian, langkah eksperimen tetap diperlukan untuk proses kalibrasi dan verifikasi.

Tidak seperti pada bidang aerodinamik dan hidrodinamik yang telah jauh lebih berkembang, aplikasi model aliran turbulen di bidang hidraulik masih banyak yang menggunakan penyederhanaan dalam memperhitungkan distribusi kecepatan aliran, dimana dalam hal ini

digunakan konsep persamaan rata-rata (*averaging equation*) dan integrasi kedalaman (*depth integration*) pada persamaan pengatur aliran. Penggunaan konsep ini telah memberikan kontribusi yang signifikan di bidang hidraulik. Dibandingkan dengan model non turbulen, hasil model turbulensi menunjukkan peningkatan akurasi yang signifikan dalam hal besar kecepatan arus, pola arus dan sebaran konstituen aliran. Model turbulen dapat mensimulasikan pola sirkulasi arus dan terbentuknya vorteks pada zona sirkulasi.

Keberadaan belokan di sungai akan menimbulkan pengaruh pada perilaku aliran yang melewatinya. Perubahan perilaku aliran yang direpresentasikan dalam kecepatan aliran dan struktur turbulensi ini akan menimbulkan perubahan pula pada distribusi sedimen dan perubahan dasar. Aliran sekunder yang digerakkan akibat perubahan alur sungai, telah dianggap terutama bertanggung jawab untuk mendistribusikan kembali atau menggeser aliran primer. Aliran sekunder yang digerakkan oleh kelengkungan disebabkan oleh perbedaan dalam gaya sentrifugal antara lapisan atas dan bawah yang melengkung mengalir. Bagaimanapun, topografi lapisan memainkan peran penting dalam pergeseran aliran. Banyak peneliti menunjukan bahwa inti dari kecepatan maksimum tetap berada di tepi dalam belokan, sampai mencapai bagian perlintasan di kelengkungan rendah dengan dasar datar. Secara kualitatif, inti aliran primer maksimum akan secara bertahap melintasi garis tengah sungai dan bergeser ke arah tepi luar belokandimana kelengkungan belokan berkembang menjadi lebih besar dan terjadi perubahan dasar saluran. Fenomena alam ini tidak bisa diamati secara langsung. Salah satu metode untuk menyederhanakannya adalah dengan pemodelan fisik maupun numerik. Pemodelan ini sangat membantu dalam memvisualisasikan baik gejala-gejala alam ataupun respon yang diberikan dari fenomena-fenomena alam tersebut.

Penelitian mengenai perilaku aliran pada belokan saluran telah banyak dilakukan, namun kurang untuk detail struktur tiga dimensi dari turbulensi aliran dan interaksinya. Kebanyakan dari penelitian tersebut bersifat eksperimental dan fokus pada masalah gerusan dan superelevasi aliran, sementara itu untuk struktur turbulensi banyak digunakan pendekatan dua dimensi rata rata kedalaman.Untuk itu dalam penelitian ini akan digunakan model aliran 3-dimensi dengan non-linier κ-ε turbulent model untuk mengkaji detail struktur tiga dimensi dari turbulensi aliran dan interaksinya dengan dasar saluran.

KEUNTUNGAN PEMODELAN NUMERIK

Terlepas dari hasil yang tepat dan pemahaman yang jelas tentang fenomena aliran; pendekatan eksperimental memiliki beberapa kelemahan seperti pengumpulan data yang sulit dan data dapat dikumpulkan untuk sejumlah titik yang terbatas karena keterbatasan pengoperasian instrumen; model biasanya tidak pada skala penuh dan perilaku aliran tiga dimensi atau beberapa struktur turbulen kompleks yang merupakan sifat dari aliran saluran terbuka tidak dapat ditangkap secara efektif melalui eksperimen. Jadi dalam keadaan ini, pendekatan komputasi dapat diadopsi untuk mengatasi beberapa masalah ini dan dengan demikian menyediakan alat yang sesuai. Dibandingkan dengan studi eksperimental; metodologi komputasi dapat diulang, dapat mensimulasikan pada skala penuh; dapat menghasilkan aliran dengan mempertimbangkan semua poin data & terlebih lagi dapat mengambil tantangan teknis terbesar yaitu; prediksi turbulensi. Struktur turbulen yang kompleks seperti sel aliran sekunder, vortisitas, tegangan Reynolds dapat diidentifikasi dengan pemodelan numerik secara efektif, yang sangat penting untuk penyelidikan aliran energi di aliran saluran terbuka. Banyak peneliti dalam beberapa abad terakhir telah memodelkan aliran saluran terbuka secara numerik dan telah berhasil divalidasi dengan hasil eksperimen.

UCAPAN TE|RIMA KASIH

Penulis ingin mengucapkan terima kasih kepada Badan Penelitian dan Pengabdian kepada Masyarakat (BPPM-FTUB) yang telah mendanai penelitian ini dan juga kepada International River Interface Cooperative (iRIC) yang telah menyediakan alat untuk simulasi.

DAFTAR PUSTAKA

1. Constantinescu, G., M. Koken, and J. Zeng (2011), *The structure of turbulent flow in an open channel bend of strong curvature with deformed bed: Insight provided by detached eddy simulation*, Water Resour. Res., 47, W05515, doi:10.1029/2010WR010114.
2. José Rodolfo Machado de Almeida1, José Junji Ota, (2020), *Comparative study between turbulence models in curved channels*, RBRH vol.25 Porto Alegre
3. A. Gholami, H. Bonakdari, A.A. Akhtari, (2016), *Developing finite volume method (FVM) in numerical simulation of flow pattern in 60° open channel bend*, Journal of Applied Research in Water and Wastewater, 3 (1), 193-200.
4. K. Blanckaert and W. H. Graf, (2001), *Experiments on flow in an open-channel bend. Mean flow and turbulence*, J. Hydraul. Eng. https://doi.org/10.1061/(ASCE)0733-9429(2001)127:10(835) 127, 835.
5. K. Blanckaert and H. J. de Vriend, (2004), *Secondary flow in sharp open-channel bends*, J. Fluid Mech. https://doi.org/10.1017/S0022112003006979 498, 353.
6. K. Blanckaert, (2004), *Turbulence characteristics in sharp open-channel bends*, J. Fluid Mech. https://doi.org/10.1017/S0022112003006979 498, 353.
7. Wolfgang Rodi, (2017), *Turbulence Modeling and Simulation in Hydraulics: A Historical Review*, Journal of Hydraulic Engineering.
8. Sukhodolov, A. N. (2012), *Structure of turbulent flow in a meander bend of a lowland river*, Water Resour. Res., 48, W01516, doi:10.1029/2011WR010765.
9. Jyotismita Taye, Jyotirmoy Barman, Mahesh Patel & Bimlesh Kumar (2019): *Turbulent characteristics of sinuous river bend*, ISH Journal of Hydraulic Engineering, DOI: 10.1080/09715010.2019.1629843
10. Xiaohui Yan · Colin D. Rennie · Abdolmajid Mohammadian, (2020), *A three-dimensional numerical study of flow characteristics in strongly curved channel bends with different side slopes*, Environmental Fluid Mechanics, https://doi.org/10.1007/s10652-020-09751-9

11. Ichiro Kimura, (2020), Solver Manual NaysCUBE v 3.1, I-RIC, Japan
12. Al-Fadhli, Ishraq Hamdan, (2013), *Turbulence characteristics in unsteady and non-uniform flows and their impacts on the incipient motion of sediment transport-water resources-river mechanics*, Master of Engineering - Research thesis, School of Civil, Mining and Environmental Engineering, University of Wollongong,. https://ro.uow.edu.au/theses/3907

Flow and sediment transport in a sharp river bend using a 3D-RANS model

Dian Sisinggih[1], **Sri Wahyuni**[1], **Adenan Rasyid**[2]

[1]Department of Water Resources Engineering, Universitas Brawijaya, Indonesia
[2]Graduate Student of Water Resources Engineering, Universitas Brawijaya, Indonesia

*Email: singgih@ub.ac.id

Abstract. Flow dynamics and sediment transport in a river bend have recently been studied using experimental and numerical investigations. A three-dimensional numerical modeling model named NaysCUBE was used in this study to describe the flow pattern and process of sediment transport in a sharp river bend as a complement to the prior work of the physical hydraulic model. The model uses the RANS equation to simulate flow where a fully complex 3D flow is governed. Despite the limitations of the RANS model, NaysCUBE well reproduces the flow pattern and turbulence phenomena in a movable bed channel with sharp-curvature. Compared with data from a prior experiment, the morphological adjustment is simulated sufficiently. The three-dimensional flow structures are useful for determining the appropriate countermeasures for local scouring and river bank protection.

Keywords: sediment, river bend, turbulent, NaysCUBE, RANS

Introduction

Flow distribution in channel bends is important in hydraulic and environmental engineering because it is used to proportion river channel sections, optimize bank protection options, design ecological and environmental development methods etc. Channel bends have quite distinct flow patterns and structures than straight channels. Flow of water and sediment transport in a sharp river bend are of significant interest in river engineering. Whether there is a steady or unsteady flow, bed erosion occurs at a sharp river bend, causing significant damage to infrastructure along the rivers. [1,2,3].

River bends have a complex three-dimensional flow field. A strong main secondary current dominates the flow structure. It produces higher shear stress in the bottom and bank, resulting in erosion along the outer bank. An imbalance of inertial forces and transverse pressure gradients causes a helical flow to form. High-velocity flow is convected towards the outer bank and pushed into the river bed, resulting in high-shear stresses and erosion potential. The eroded sediments are transported across the river section and deposited on the inner bank as the flow is recirculated from the outer bank to the inner bank. [4].

As computer power has expanded, modeling turbulent flows in bends and meandering channels has become more common. The unsteady three-dimensional Navier-Stokes equations are not directly solved in existing current numerical models for flow in open channels and rivers. It is too expensive to apply direct numerical simulations (DNS) to actual water flows at high Reynolds numbers [4].

Reynolds-Averaged Navier-Stokes (RANS) equations are utilized in most models due to their computational efficiency and performance. Here, the Reynolds stress tensor must be represented using turbulence model equations [5,6].

The flow pattern of meandering rivers is highly complicated, having distinct characteristics at bends that are not seen along straight channel. Such flow fields can be effectively predicted using a numerical model. Several isotropic RANS turbulence models, notably the kappa-epsilon model, play a prominent role in real applications. Flows in curved open channel flumes, natural river flows, and delta morphodynamics have all been predicted using these models. RANS models often accurately represent the fundamental flow structure [7,8,9]. The good performance is likely due to the fact that the flow topologies in those test examples are relatively basic, with only one major circulation in each cross section. Despite their widespread use, the RANS models' capacity to forecast complex flow configurations has yet to be tested. In many occasions, RANS model predictions were proven to be inaccurate; nonetheless, numerous issues, such as influencing variables, inherent causes, and the level of inaccuracy under various circumstances, remain unsolved [10,11,12,13].

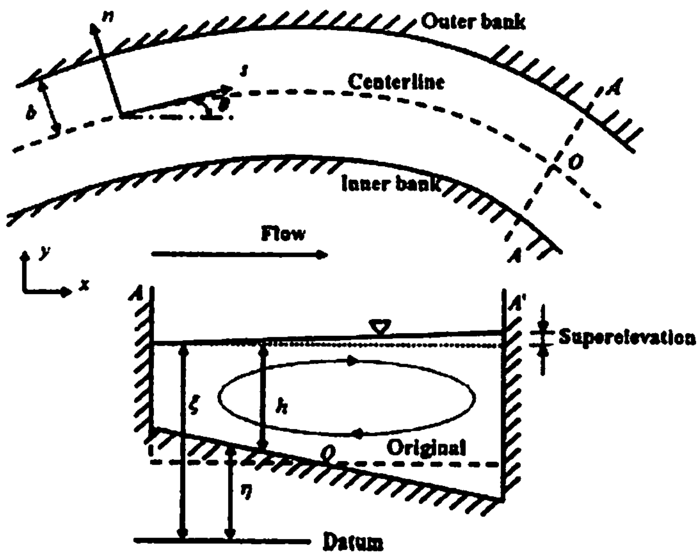

Figure 1. Definition and sketch of river meander [14]

This paper describes the performance of the application of the RANS model (NaysCUBE) for investigation of flow behavior in sharp river bends. This result is complementary to prior physical hydraulic model experiments.

Material and Methods

2.1 Description of the study

A physical hydraulic model was developed in the laboratory to analyze the flow behavior in river bends with a moveable bed. A three-dimensional numerical model with non-linear-turbulent models is also used to describe the mechanism of developing and decaying turbulence in the flow.

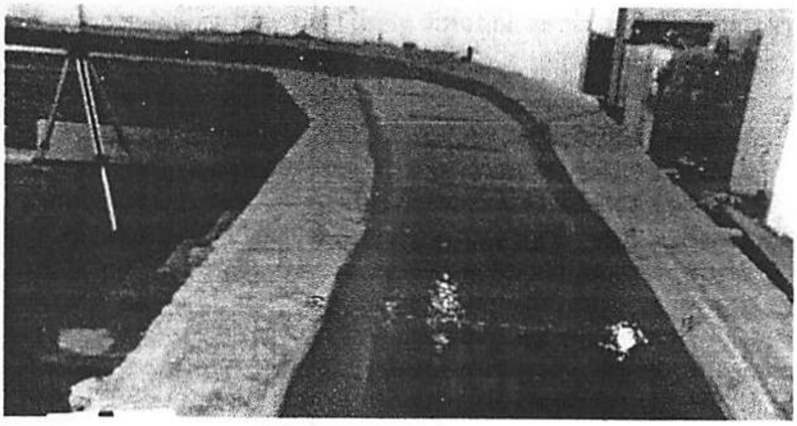

Figure 2. Physical hydraulic model of an alluvial channel with a sharp bend (r/B < 2.5)

Table 1. List of the experiment data used

Data Type	Description	Prototype	Physical hydraulic model
River reach	A sharp bend with a radius (r) of 75m with a length of 150m	Bottom width = 12m Top width = 35m Average slope 0.0025 Flow regime= subcritical Froude ~ 0.4 Averaged velocity = 1.4 m/s	Undistorted 1:20
Hydrology data	Q_{30}	53.67 m^3/s	30 l/s
Sediment data	uniform	ρ_s = 2.680 ton/m^3 d_{50} = 0.15 mm d_{90} = 0.30 mm	ρ_s = 1.538 ton/m^3 d_{50} = 0.55 mm d_{90} = 2.25 mm
Numerical Model	RANS - kappa epsilon model (NaysCUBE) with Meyer Peter Muller for bed sediment transport		

2.2. Numerical Simulation

The International River Interface Cooperative (IRIC) created the solver for Reynolds-Averaged Navier-Stokes (RANS) equations, which is freely available on the internet. We performed numerical computations using iRIC Software (NaysCUBE) for 3D flow analysis to further investigate the bed mrophological changes and flow pattern as yielded by the experiment.

2.2. Summary of NaysCUBE

Professor Ichiro Kimura of Toyama University developed NaysCUBE, a fully unsteady three-dimensional solution for open channel flow and bed morphological changes [5,15]. The method solves the fundamental equations of three-dimensional flow while considering nonhydrostatic water pressure and high vertical accelerations and velocities. The following are the NaysCUBE formulas. In the Cartesian coordinate system, Formula (1) represents the equation of continuity. The equations of motion in three dimensions are shown in Formula (2). The kappa-epsilon equation is shown in formula (3) and (4), respectively. The formula is transformed into a generalized curvilinear coordinate system.

NaysCUBE calculates the turbulent-flow field using diffusion terms, and the kappa-epsilon model was applied to the turbulent-flow field.

$$\frac{\partial U^i}{\partial x^i} = 0 \tag{1}$$

$$\frac{\partial U^i}{\partial x^i} + \frac{\partial U^i U^j}{\partial x^j} = G^i - \frac{1}{\rho}\frac{\partial p}{\partial x^i} + \frac{\partial\left(-\overline{u^i u^j}\right)}{\partial x^j} + \nu\frac{\partial^2 U^i}{\partial x^i \partial x^j} \tag{2}$$

$$\frac{\partial k}{\partial t} + \frac{\partial k U^j}{\partial x^j} = -\overline{u^i u^j}\frac{\partial U^i}{\partial x^j} - \varepsilon + \frac{\partial}{\partial x^j}\left\{\left(\frac{\nu_t}{\sigma_k} + \nu\right)\frac{\partial k}{\partial x^j}\right\} \tag{3}$$

$$\frac{\partial \varepsilon}{\partial t} + \frac{\partial \varepsilon U^j}{\partial x^j} = -C_{\varepsilon 1}\frac{\varepsilon}{k}\overline{u^i u^j}\frac{\partial U^i}{\partial x^j} - C_{\varepsilon 2}\frac{\varepsilon^2}{k} + \frac{\partial}{\partial x^j}\left\{\left(\frac{\nu_t}{\sigma_\varepsilon} + \nu\right)\frac{\partial \varepsilon}{\partial x^j}\right\} \tag{4}$$

where, x^i: spatial coordinates; t: time; U^i: flow velocity; p: pressure; u^i: turbulent velocity; ν :kinematic viscosity coefficient; ρ: fluid density; k: turbulent kinematic energy; ε: dissipation rate of turbulent kinematic energy; ν_t: turbulent kinematic viscosity coefficient; G^i: gravity acceleration; $\overline{u^i u^j}$: Reynolds stress tensor.

2.4. Other Conditions

The prototype river was replicated using a numeric model. The computational mesh was constructed at a range of 1.0-2.0 m in grid size, as illustrated in Figure 3 below. Following the experiment conducted in the physical model hydraulic, the flood hydrograph was used as the upstream boundary condition, while the water level was used as the downstream boundary condition. In both the upstream and downstream boundaries, there was no sand feeding. $n = 0.025\ m^{1/3}/s$ was chosen as the general roughness coefficient.

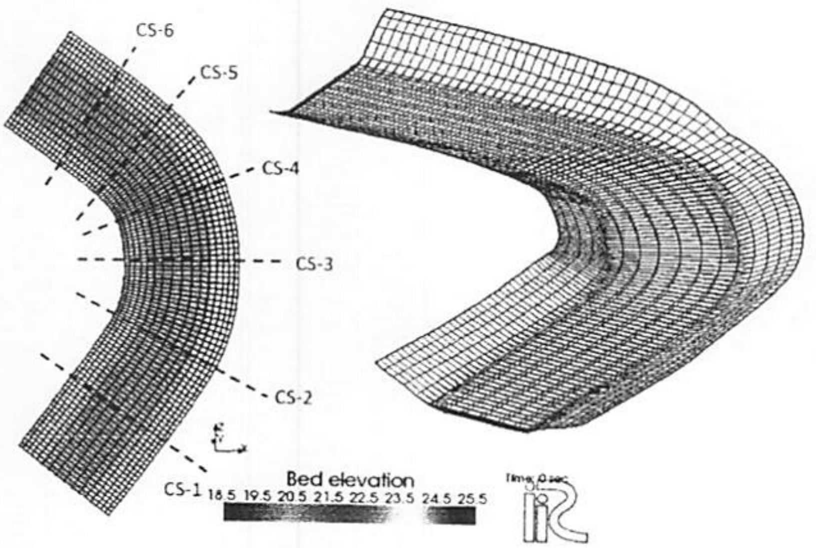

Figure 3. The perspective view of river bend in the study and its computational mesh with selected sections which observed in physical model.

Result and Discussion

The numerical simulation was run for 3 hours of flood according to the hydrological data. With a few significant exceptions, the findings show that three-dimensional flow patterns match the conceptual model of flow across river bends (Figure 4). The summary of findings is as outlined below:

3.1 Velocity Patterns

The pattern of depth-averaged mean velocity vectors best represents a general summary of the velocity field in the meander bend. Figure 4 shows that the flow pattern at the entry is asymmetric and tilted toward the upstream bend's outer bank. The flow pattern at the entrance is asymmetric and inclined outwards, as seen in Figure 4. It progressively turns into a rather symmetrical pattern downstream from the apex. Here, the most important outcome is the maximum velocity in both horizontal and vertical locations, as seen in Figures 4 and 5.

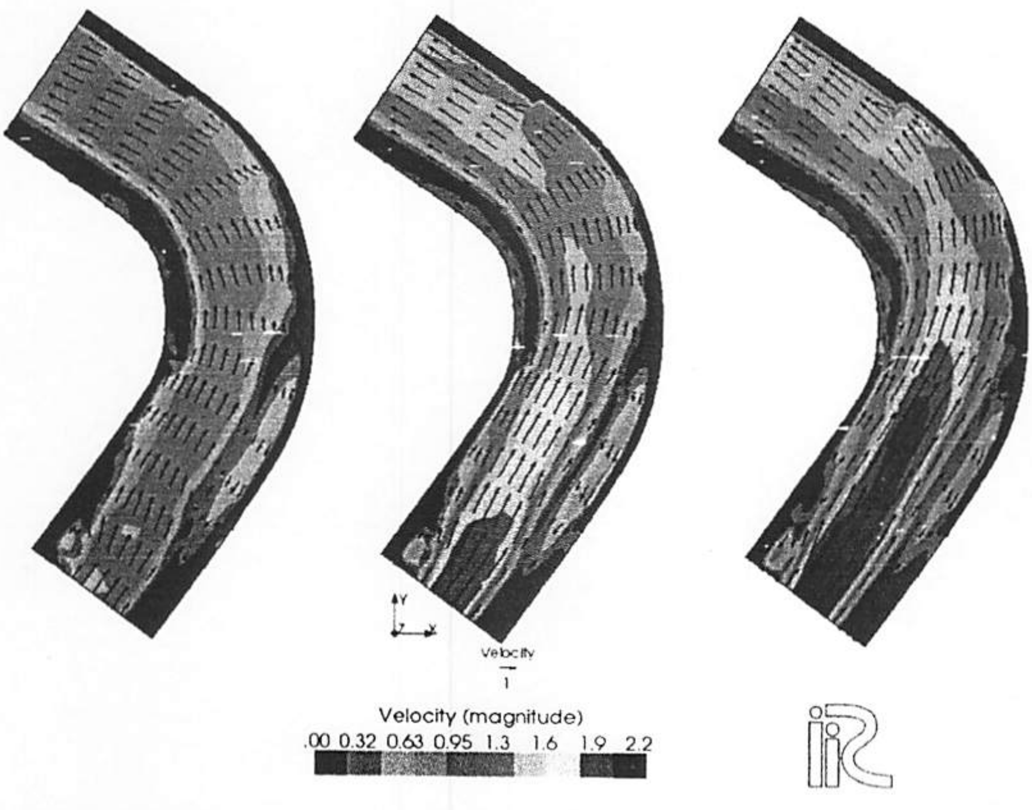

Figure 4. Contour and flow velocity vector at the bottom, mid and surface channel respectively.

3.2 Primary Flow Overview

Additionally, depth averaged patterns gave data that showed flow field characteristics in the horizontal plane. In the vertical plane, the profile of flow velocity is depicted in Figure 5. The highest flow velocity is convergent and near the surface at the entrance. The flow velocity near the right bank is slowly lowered to virtually zero at the center of the flow in cross-section 1 and cross-section 2. This is due to the presence of a minor separation zone near the outer bank. Following the pattern of depth averaged velocity vectors, the velocity converged to the center when approaching the curve and after the bend. According to Figure 5, CS 4, 5, and 6 show good agreements between measurement and simulation, but CS 1, 2, 3, the output model deviates slightly from measured data, particularly for the right bank velocities. Since the propeller velocity meter was used, there is a limit to measuring the main velocity only at a certain depth. The development of secondary flow close to the right banks caused the existence of reverse flow (cross-section 1 to cross-section 3), which appears to be the reason for the incorrect reading in the measurements. Overall, the numerical results are also supported by observed values (red dots) from the experiment in the physical model.

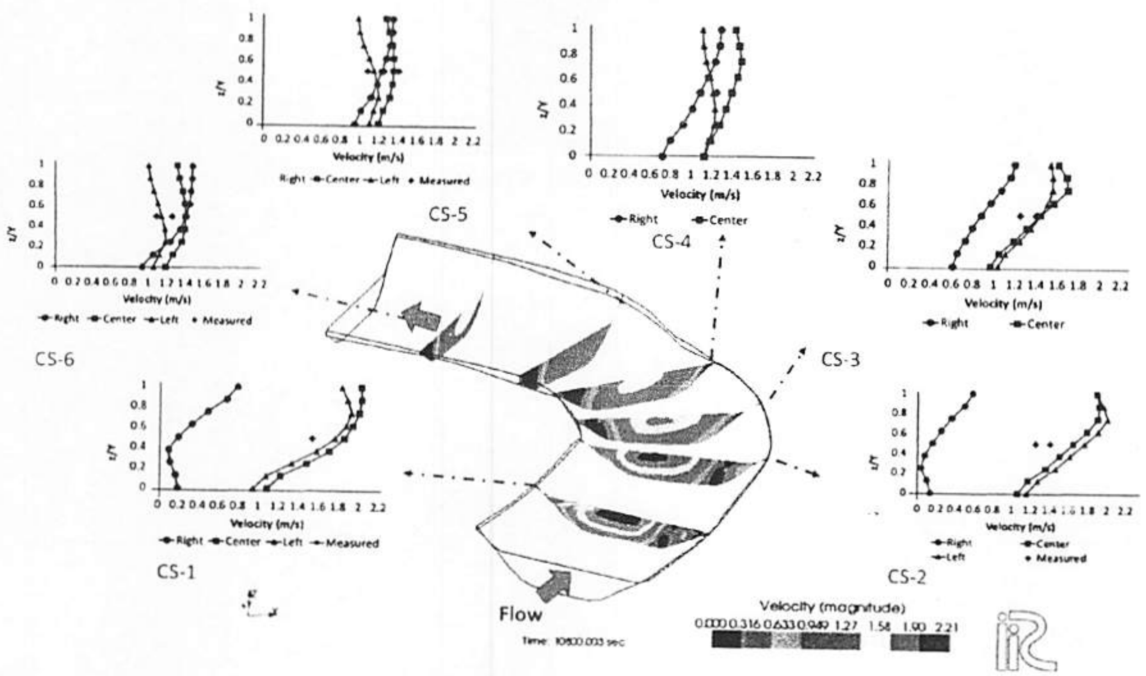

Figure 5. Perspective views in three dimensions and profile cross-section plots with overlaid velocity for each section measured in the physical model.

3.3 Pattern of Secondary Flow

In meander bends, secondary flow is a unique feature of water flow. Velocity vectors show the patterns of secondary flow and how they develop over the meander curve. (Figure 6). Flow patterns in cross sections downstream from the bend apex also suggest the development of a secondary counter-rotating cell towards the outer bank. The counter-rotating cell is found in one-third of the water depth, with the greatest interaction with the main cell of the secondary flow towards the outer bank's bed. There were strong reverse flows in the cross-section 1 to cross-section 3. Similarly, the minor one also expands along the inner bank before decaying downstream (fourth, fifth, and sixth cross-sections).

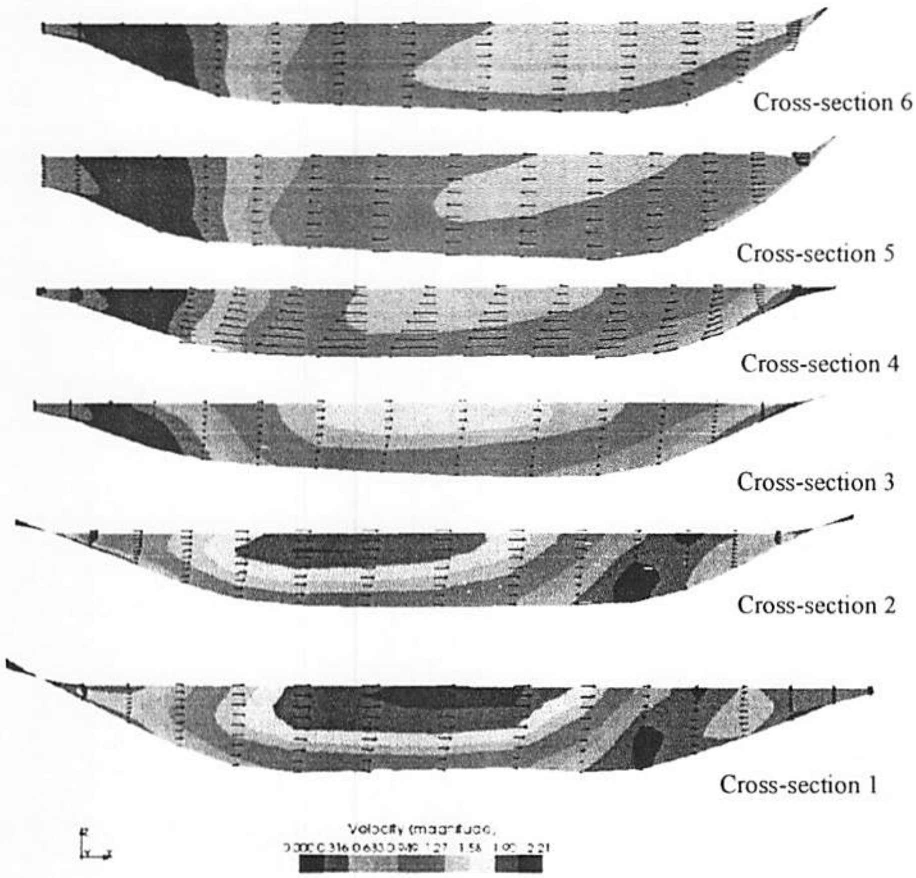

Figure 6. Selected cross-section plots measured in a physical model, showing the contour of velocity magnitudes with overlay vertical velocity vectors.

3.4 Pattern of Turbulence

Figure 7 depicts the distribution of turbulent kinetic energy (TKE in m^2/s^2) in the section featuring the turbulence structure. The convergence of the velocity vectors in the centre part corresponds to the area with high k, which intensifies turbulence as described in the previous paragraphs. TKE reaches its maximum value at the center of the flow depth, which varies with channel alignment, and then steadily decreases beyond the bend apex. Figure 8 depicts the patterns of eddy viscosity coefficients in the river bend, which correspond to turbulent kinetic energy.

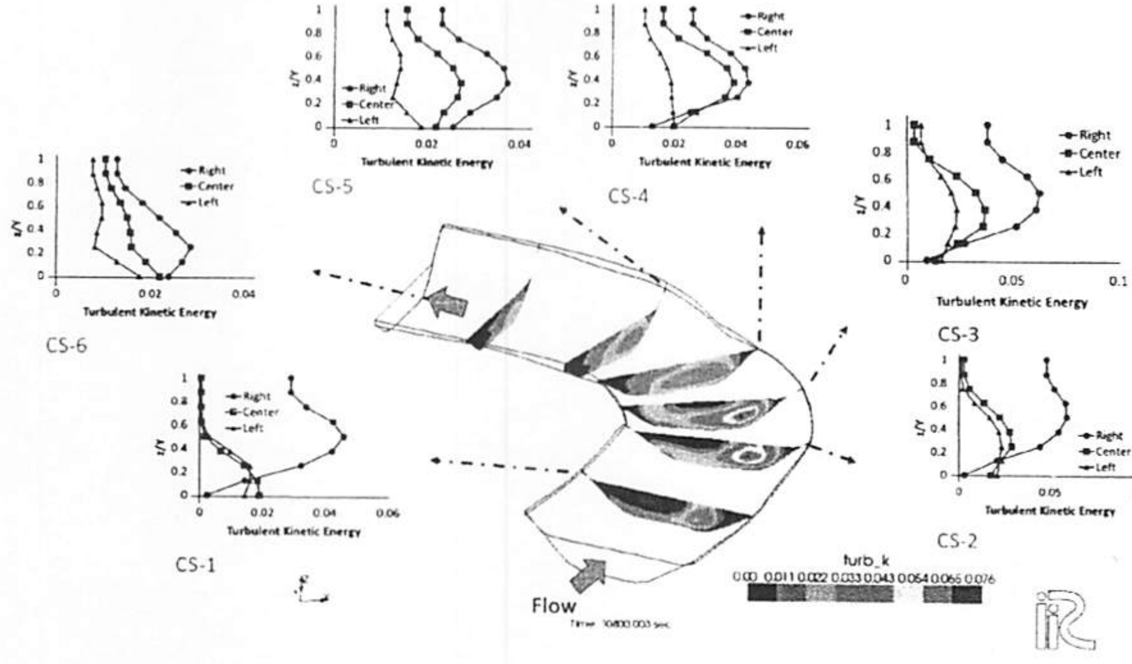

Figure 7. Perspective views in three dimensions and profile cross-section plots showing turbulent kinetic energy magnitudes.

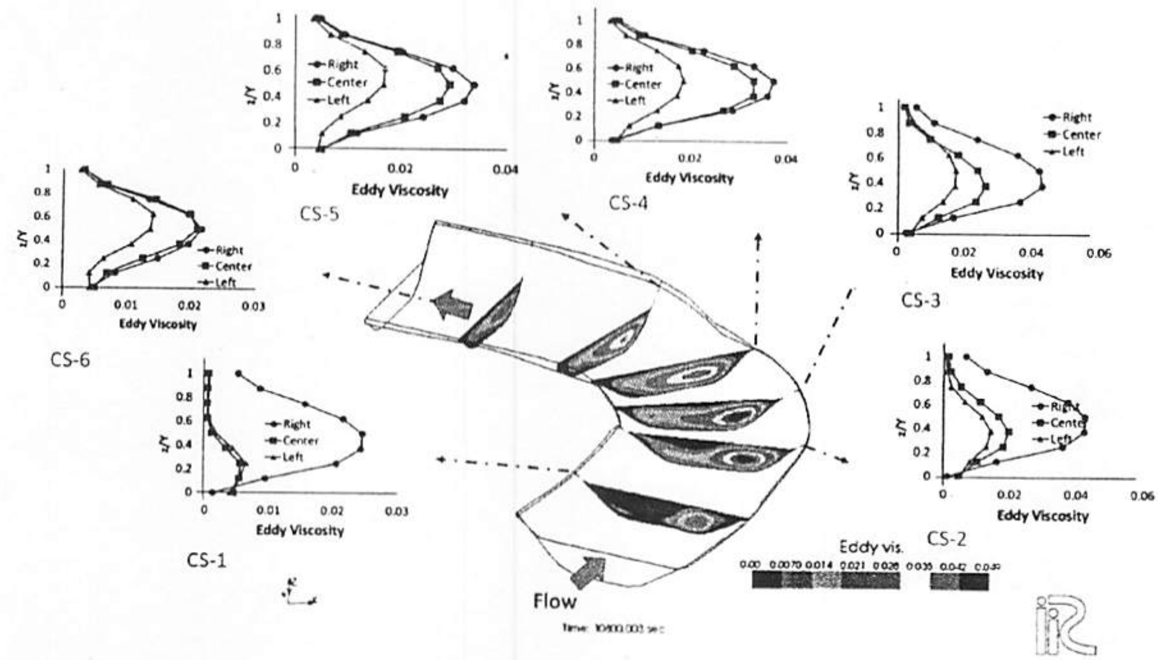

Figure 8. Perspective views in three dimensions and profile cross-section plots eddy viscosity magnitudes.

3.5 Bed morphology and Sediment transport

The local scour developed downstream due to no sediment supply from upstream. During the early stages of the studies, scour was rapid, but it slowed when the local scour approached equilibrium. With increasing mixing and turbulence intensities, the length and width of the local scour grew. A deep and centralized local scour progressively formed towards the outer bend apex and downstream, according to a physical hydraulic model and numerical results. The bed material was carried fast due to the substantial mixing activity downstream of the river bend. With no additional material, the local scour grew deeper and larger until stabilizing. The final bed adjustments are shown in Figure 9. In summary, the RANS model with a selected MPM transport formula gave smaller morphological changes compared with physical experiments in Figure 10.

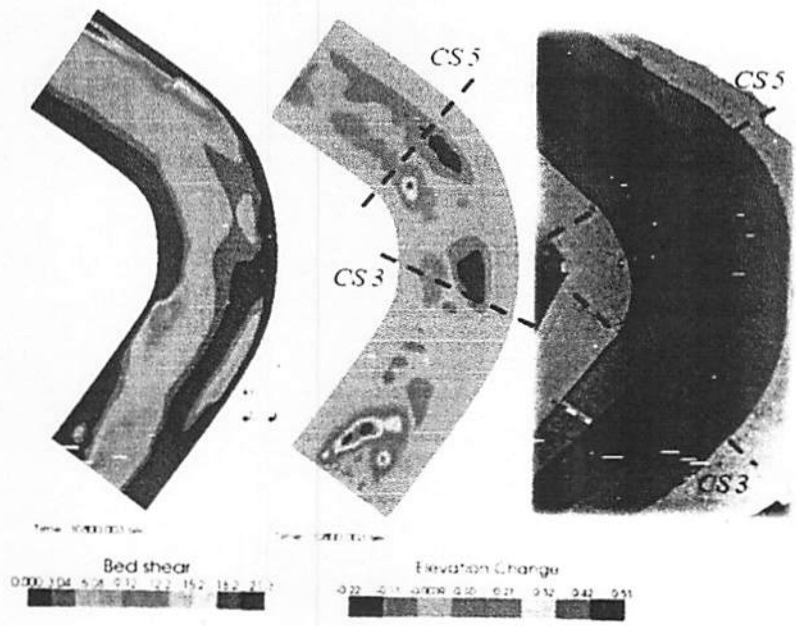

Figure 9. Magnitude (contour) of bed shear stress and riverbed migration patterns (experiment and numerical)

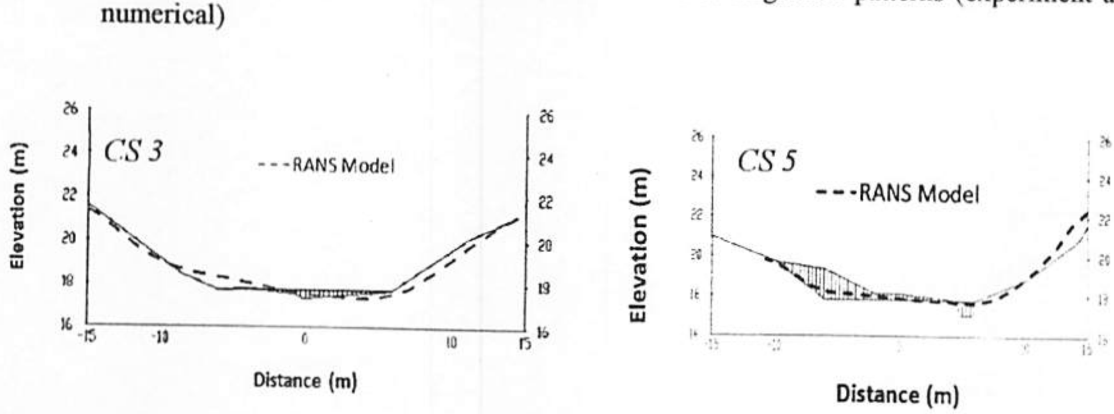

Figure 10. Comparison bed changes between the RANS model and the physical hydraulic model (CS 3 and CS 5).

Conclusions

The study of flow patterns and sediment transport in meandering channels is extremely useful in river engineering, river network and channel design, as well as river mechanics and stream evolution. The performance of the RANS model (NaysCUBE) for investigating flow dynamics at severe river bends is described in this study. This finding might be valuable in conjunction with previous physical hydraulic model studies. The following are the findings:

- NaysCUBE well reproduces the flow pattern and turbulence phenomena in a movable bed channel with sharp-curvature.
- Compared with data from a prior physical experiment, the morphological adjustment is insufficient simulated for the selected MPM transport formula. It is recommended to check another bed load formula and re-calibrate the sediment parameters in the model.
- The three-dimensional flow structure is useful for determining the appropriate countermeasures for local scouring and river bank protection.

In order to get better morphological parameters, an experiment with a physical hydraulic model should be installed with proper equipment for detecting the motion of bed particles. Instantaneous three-dimensional velocities measurement device should be installed too.

Acknowledgment

The authors would like to acknowledge the Faculty of Engineering, the University of Brawijaya, Research and Community Service (BPPM-FTUB) for sponsorship of this study and also to the iRIC team for providing tools for simulation.

References

[1] Thorsten Stoesser, Nils Ruether, Nils Reidar Boe Olsen 2010 *Calculation of primary and secondary flow and boundary shear stresses in a meandering channel Advances in Water Resources 33 158–170*

[2] Sukhodolov, A. N. 2012 *Structure of turbulent flow in a meander bend of a lowland river Water Resour. Res.*, 48 W01516, doi:10.1029/2011WR010765.

[3] Jyotismita Taye, Jyotirmoy Barman, Mahesh Patel & Bimlesh Kumar 2019 *Turbulent characteristics of sinuous river bend J. Hydraulic Engineering*, DOI: 10.1080/09715010.2019.1629843

[4] Okamoto, Y. & Nishio, J. & Kanda, K. & Michioku, Kohji & Nakamura, F. & Kubo, H.. 2020 *Study on riverbed variation management by groin at a river confluence associated with the barrage water. Proc. 10th Conference on Fluvial Hydraulics* 10.1201/b22619-92

[5] Taeun Kang, Ichiro Kimura and Yasuyuki Shimizu 2020 *Numerical simulation of large wood deposition patterns and responses of bed morphology in a braided river using large wood dynamics model Earth Surf. Process. Landforms* 45, 962–977 DOI: 10.1002/esp.4789

[6] Comp Mehrzad Shams, Goodarz Ahmadi, Duane H. Smith 2002 *Computational modeling of flow and sediment transport and deposition in meandering rivers Advances in Water Resources* 25 689–699.

[7] Jian-yin Zhou, Xue-jun Shao, Hong Wang, Dong-dong Jia 2017 *Assessment of the predictive capability of RANS models in simulating meandering open channel flows. J. Hydrodynamics.*,29(1):40-51 DOI: 10.1016/S1001-6058(16)60714-X

[8] Chang Geun Song, Il Won Seo, Young Do Kim 2012 *Analysis of secondary current effect in the modeling of shallow flow in open channels Advances in Water Resources* 41 29–48

[9] İnci Güneralp, Jorge D. Abad, Guido Zolezzi, Janet Hooke 2012 *Advances and challenges in meandering channels research, Geomorphology 163*–164 1–9

[10] W. Ottevanger, K. Blanckaert, W.S.J. Uijttewaal, 2012 *Processes governing the flow redistribution in sharp river bends, Geomorphology* 163–164 45–55

[11] Frank L. Engel, Bruce L. Rhoads 2012 *Interaction among mean flow, turbulence, bed morphology, bank failures and channel planform in an evolving compound meander loop Geomorphology* 163–164 70–83

[12] James D. Riley, Bruce L. Rhoads 2012 *Flow structure and channel morphology at a natural confluent meander bend, Geomorphology* 163–164 84–98

[13] Jianchun HUANG, Blair P. GREIMANN, and Timothy J. RANDLE 2014 *Modelling of meander migration in an incised channel Int. J. Sediment Research* 29 441-453

[14] Thorsten Stoesser, Nils Ruether, Nils Reidar Boe Olsen 2010 *Calculation of primary and secondary flow and boundary shear stresses in a meandering channel Advances in Water Resources 33 158–170*

[15] Kimura, I 2020 NaysCUBE ver 3.1 Solver Manual iRIC Software