COMET editorial system | Tue 8th of August 2023 | 15:42 cet

Welcome **Dr. Tuswan Tuswan**
Pending tasks: **0** | My info | Logout

AUTHOR

START NEW SUBMISSION
waiting for upload
waiting for minor revision
waiting for major revision
waiting for techn. reupload
ALL SUBMISSIONS

Comet v3.0
build date: (ongoing)

Details for: **BROD.0074**

GENERAL | REVIEWS | HISTORY | UPLOADS

MANUSCRIPT

DECISION: **REVISION NEEDED**

Dear Dr. Tuswan, please format your manuscript according to the template of the journal Brodogradnja and resubmit. It is important for authors and the journal to cite appropriate papers published earlier in Brodogradnja if they are of possible interest. You are strongly encouraged to include at least three references to papers previously published in Brodogradnja, preferably in the last two years. Kind regards, Professor Nastia Degiuli, Editor in Chief PhD Andrea Farkas, Executive editor for data base administration PhD Ivana Martić, Executive editor for similarity checking

DECISION: **REVISION NEEDED**

Dear Dr. Tuswan, please format your manuscript according to the template of the journal Brodogradnja (https://brodogradnja.fsb.hr/wp-content/uploads/sites/25/2022/09/BRODOGRADNJA_Template.doc) and resubmit. Kind regards, Professor Nastia Degiuli, Editor in Chief PhD Andrea Farkas, Executive editor for data base administration PhD Ivana Martić, Executive editor for similarity checking

DECISION: **REVISION NEEDED**

Dear Dr. Tuswan, please format the references according to the template of the journal Brodogradnja (https://brodogradnja.fsb.hr/wp-content/uploads/sites/25/2022/09/BRODOGRADNJA_Template.doc). Please note that year of publication should not be within the brackets, and please remove "&" between the last two authors' names. Kind regards, Professor Nastia Degiuli, Editor in Chief PhD Andrea Farkas, Executive editor for data base administration PhD Ivana Martić, Executive editor for similarity checking

DECISION: **PRESCREENING PASSED**

Dear Dr. Tuswan, Your manuscript has been successfully submitted and is presently being given full consideration for publication in the journal Brodogradnja. Please note that for all papers submitted after 31 December 2019 an Article Publishing Fee applies. The APF amounts to 350 Euros and has to be paid after article/paper acceptance before publication (see Submission guides for authors).

DECISION: **ACCEPTED FOR PUBLICATION AFTER MAJOR REVISION**

****
* Overall rating:
57

* Categorisation
Scientific paper

* Does the paper's title convey the content of the paper?
Yes

* Is the paper's abstract concise and informative while conveying the content of the paper?
Yes

* The information presented was new.
1 - Fails by a small amount

* The subject addressed in this paper is worthy of investigation.
2 - Succeeds by a small amount

* The conclusions were supported by the data.
0 - Fails by a large amount

* The completeness of literature review.
2 - Succeeds by a small amount

* Is the English of the paper suitable for publication?
2 - Succeeds by a small amount

* Are you aware of prior publication of this work?
No

* Is there a financial or other conflict of interest between your work and that of the authors?
No
****

Please fix the following issues in the manuscript: 1. Fig 1 does not contribute to depicting the hull form. Fig 2 should be kept, but "body plan" should be larger, as it is the main information to reproduce the model and results. 2. "by comparing the hydrostatic values" is not a phrase. Does this mean the authors only compared the displacement? In any case, please be specific when describing the preparation of the model. In addition, "validity" only applies to the results, and here you can use "verification". 3. "numerical setup was initially carried out by comparing the computational results" -> initially validated 4. Please update: hw/T of 2.24 -> hw/T = 2.24 (and similar phrases). 5. When using L as length on the waterline, please denote it as L_wl. 6. Please remove the theory part in Section 2.5, because it is widely accepted method and does not contribute to anything. Only turbulence models, time stepping, and Table 4 are important. 7. Y+ is dangerously close to the transitional range, how do authors explain this? 8. Validation test should be made using multiple meshes (especially since your meshes in different cases vary a lot!). You can use well known procedure from doi:10.1115/1.2960953 9. The validation underpredicts the experimental results for Froude numbers in which wavemaking is important. It means that your wavemaking procedure is not converging well enough. Fig 10. also brings up this uncertianty, since you can see the coarse wetted regions. For this reason, explaining point 8 is even more important! 10. In Table 6 please leave only one decimal, it is too hard to analyze it. 11. "resistance value increased to 699.69%" is also hard to understand. Compared to what initial value? 12. The authors should shown how the dynamic pressure field is changing based on the closeness of the bottom in more detail. The image of velocity Fig 20, is not important. Rather, X-Z plane centreplane cut showing dynamic pressure for various depths should be shown. This is the real information on why wavemaking changes, and the resistance force! 13. Please describe physically what happens with help of point 12, as well as your other results, instead of simply comparing the numbers and leaving the rest to readers. 14. Conclusions are wrong/not contributing. After the proper validation and physical analysis, please conclude your findings in a way engineers and researchers can see under which conditions the increase in resistance happens, and physically why. 15. Fig 24. also denotes there may be (surely there is) an issue with overset grid interpolation. please see the corners and comment on this.

DECISION: **ACCEPTED FOR PUBLICATION AFTER MINOR REVISION**

****
* Overall rating:
60

* Categorisation
Scientific paper

* Does the paper's title convey the content of the paper?
Yes

* Is the paper's abstract concise and informative while conveying the content of the paper?
Yes

* The information presented was new.
3 - Succeeds by a large amount

* The subject addressed in this paper is worthy of investigation.
3 - Succeeds by a large amount

* The conclusions were supported by the data.
3 - Succeeds by a large amount

* The completeness of literature review.
3 - Succeeds by a large amount

* Is the English of the paper suitable for publication?
2 - Succeeds by a small amount

* Are you aware of prior publication of this work?
No

* Is there a financial or other conflict of interest between your work and that of the authors?
No
****

Thank you for your manuscript. I think this study is worthy of investigation and it is well written in general. However, following points should be revised before the manuscript is considered for publication. Please find my comments and suggestions below. Best regards, 1. Abstract. Following sentence in the abstract does not read well. Please rephrase it. “Although several investigations are found in investigating the vessel behaviour in restricted water, a few research has been carried out in analyzing the different canal types and cross-sections in ship hull form.” 2. Equation (3) Please double check with the governing equation. I think the tensors (i, j, k) are missing. 3. Fig. 10. With the current colourbar range, it is not easy to see whether the hull wetted areas have proper wall y+ values (i.e. y+ >30). I suggest the authors either change the upper/lower limits of the colour map, or change the colour map style to solve this issue. 4. Boundary condition (Fig. 5-7) I can see that you used Velocity inlet for the bottoms of the tanks, while using no-slip wall for the sides. Do you have any specific reasons? Especially for the canal simulations, I think No-slip wall with velocity (i.e. moving wall) boundary condition might be a better choice. Also, you did not mention whether you applied any velocity for the no-slip wall for the side of the canal. If you did not apply the velocity on the no-slip wall for the sides (to make it a moving wall), the simulation will not represent correctly the flow around the ship and canal walls. To confirm this, can you please show me the velocity contour around the hull including the reason near the side walls? 5. Fig. 13 Please check with the caption of Fig. 13. I think “Cases I-IV” & “canal bank” should be correct. 6. Results (Fig. 13-18) Now the results are plotted against the Froude number based on the ship length. I think plotting the results with depth Froude number might be beneficial. Please consider plotting the results with depth Froude number too (maybe you can use 3D figures for both axes, i.e. normal Fr and depth Fr). Fig. 20. I am sure that you can change the spacing of the vector arrows. Now the arrows are located at their cell centres, but you can change it to uniform spacing. I am sure that the figure will look much better by doing so.

DECISION: **ACCEPTED FOR PUBLICATION AFTER MAJOR REVISION**

Thank you for considering Brodogradnja. Please see the comments of the reviewers on the revised manuscript Influence of the Canal Width and Depth on The Resistance of 750 DWT Perintis Ship Using Computational Fluid Dynamic (CFD) Simulation. I suggest you consider these comments, suggestions and questions and revise your article accordingly. For your guidance, reviewers' comments are appended below. If you decide to revise the article, please submit a list of changes or a rebuttal against each point which is being raised when you submit the revised manuscript.

DECISION: **ACCEPTED FOR PUBLICATION**

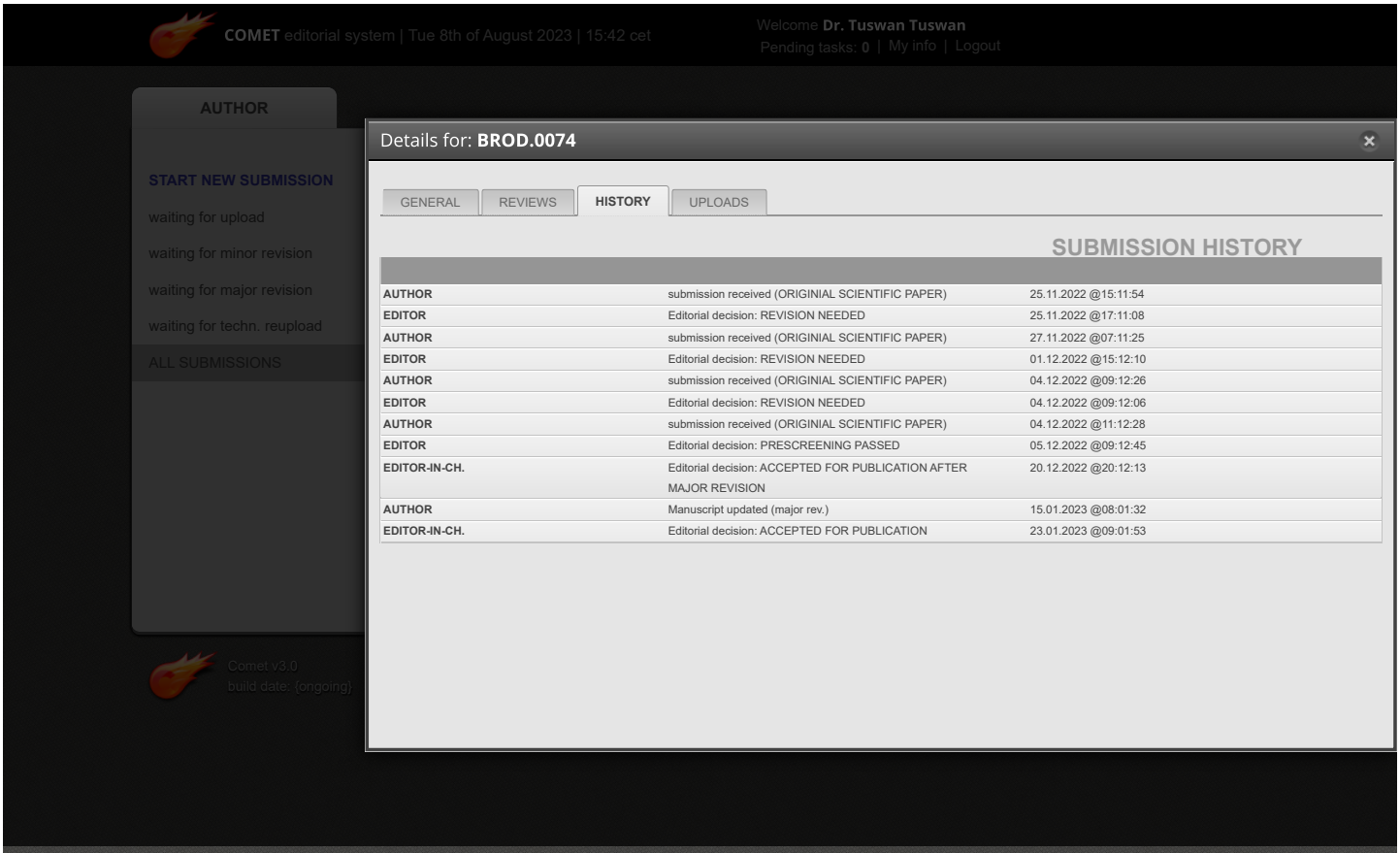
Thank you for addressing my comments. Best regards,

DECISION: **ACCEPTED FOR PUBLICATION**

the authors have fixed the manuscript for publication

DECISION: **ACCEPTED FOR PUBLICATION**

/



AUTHOR	submission received (ORIGINIAL SCIENTIFIC PAPER)	25.11.2022 @15:11:54
EDITOR	Editorial decision: REVISION NEEDED	25.11.2022 @17:11:08
AUTHOR	submission received (ORIGINIAL SCIENTIFIC PAPER)	27.11.2022 @07:11:25
EDITOR	Editorial decision: REVISION NEEDED	01.12.2022 @15:12:10
AUTHOR	submission received (ORIGINIAL SCIENTIFIC PAPER)	04.12.2022 @09:12:26
EDITOR	Editorial decision: REVISION NEEDED	04.12.2022 @09:12:06
AUTHOR	submission received (ORIGINIAL SCIENTIFIC PAPER)	04.12.2022 @11:12:28
EDITOR	Editorial decision: PRESCREENING PASSED	05.12.2022 @09:12:45
EDITOR-IN-CH.	Editorial decision: ACCEPTED FOR PUBLICATION AFTER MAJOR REVISION	20.12.2022 @20:12:13
AUTHOR	Manuscript updated (major rev.)	15.01.2023 @08:01:32
EDITOR-IN-CH.	Editorial decision: ACCEPTED FOR PUBLICATION	23.01.2023 @09:01:53

Rebuttal

Dr. Tuswan, email: tuswan@lecturer.undip.ac.id
Department of Naval Architecture, Universitas Diponegoro, Semarang, Indonesia
Jl. Prof Soedarto, SH, Tembalang, Semarang 50275, Indonesia

January 15th, 2022

Dear
Professor Nastia Degiuli
Editor-in-Chief
Brodogradnja/ Shipbuilding

Thank you for giving me the opportunity to submit a revised draft of my manuscript entitled *"Influence of the Canal Width and Depth on The Resistance of 750 DWT Perintis Ship Using Computational Fluid Dynamic (CFD) Simulation"* by Eko Sasmito Hadi, Tuswan Tuswan, Ghina Azizah, Baharuddin Ali, Muhammad Luqman Hakim, Muhammad Raaflie Caesar Putra Hadi, Muhammad Iqbal, Dian Purnamasari, Dendy Satrio for consideration for publication in the Brodogradnja/ Shipbuilding. We appreciate the time and effort you and the reviewers have dedicated to providing your valuable feedback on my manuscript. We are grateful to the reviewers for their insightful comments on my paper. We have been able to incorporate changes to reflect most of the suggestions provided by the reviewers. We have highlighted the changes in blue font colour within the manuscript. Here is a point-by-point response to the reviewers' comments and concerns.

Sincerely,

Dr. Tuswan, ST.

Reviewer 1

Please fix the following issues in the manuscript:

1. Fig 1 does not contribute to depicting the hull form. Fig 2 should be kept, but "body plan" should be larger, as it is the main information to reproduce the model and results.
   Answer: Thank you for the recommendation. We agree with your suggestion. We have revised Figure 2 with larger view of body plan. The revised Figure 2 is as follows:

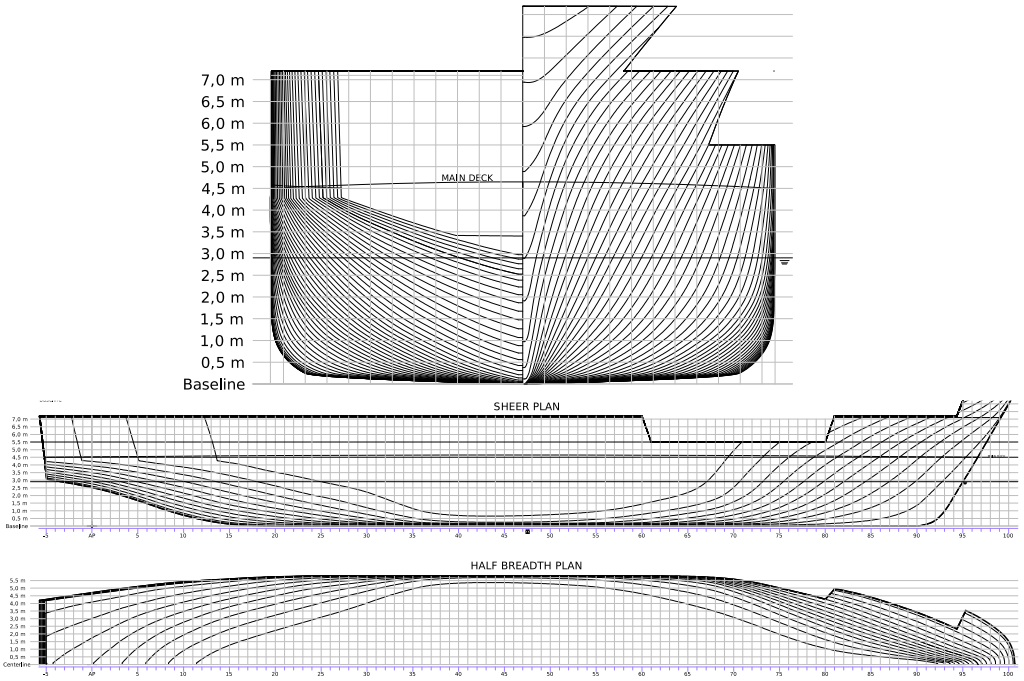


Fig. 2 750 DWT Perintis Ship lines plan.

2. "by comparing the hydrostatic values" is not a phrase. Does this mean the authors only compared the displacement? In any case, please be specific when describing the preparation of the model. In addition, "validity" only applies to the results, and here you can use "verification".
   Answer: We agree with your opinion. We have changed using term “verification”. The text has been revised as follows:
   The full-scale model was scaled with a scale factor of 1:18 using Rhinoceros and DELFTship software to perform the simulation. The model verification of the 3D hull shape was determined by comparing the principal dimension and hydrostatic values such as displacement, Cb, LCB etc.

3. "numerical setup was initially carried out by comparing the computational results" -> initially validated.
   Answer: Thank you for your suggestion. The text has been revised as follows:

The numerical setup was initially validated by comparing the computational results to the IHL towing tank experimental model test data using the total resistance value to validate the computational results

4. Please update: hw/T of 2.24 -> hw/T = 2.24 (and similar phrases).
   Answer: We have revised the phrases. The changes are highlighted in blue font colour.

5. When using L as length on the waterline, please denote it as L_wl.
   Answer: We have revised it to "Lwl". The changes are highlighted in blue font colour.

6. Please remove the theory part in Section 2.5, because it is widely accepted method and does not contribute to anything. Only turbulence models, time stepping, and Table 4 are important.
   Answer: We agree with your suggestion. We have removed the equations of RANS and volume fraction in Section 2.5.

7. Y+ is dangerously close to the transitional range, how do authors explain this?
   Answer: We thank for your suggestion. The figure has been revised as also suggested by reviewer 2 to change the upper/lower limits of the colour map and change the colour map style as follows:

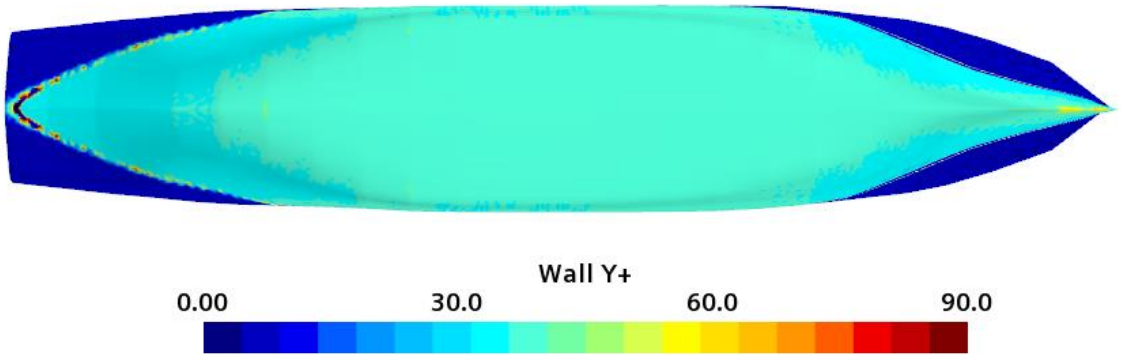


Fig. 10 y+ contour on the Perintis 750 DWT ship hull in the range of (0–90).

8. Validation test should be made using multiple meshes (especially since your meshes in different cases vary a lot!). You can use well known procedure from doi:10.1115/1.2960953
   Answer: The changes are highlighted in blue font colour. We use grid dependency studies using three grid variations as follows:
   The resulting ship grid was characterized by a concentration of cells on the free surface of the hull grid dependency studies have been made for three grids (Grid 1, Grid 2, Grid 3) with corresponding cell numbers of 3.9, 4.3, and 4.6 million. Table 5 shows the errors in the numerical results for all three grids combined with the experimental resistances. The difference between the experiment data, denoted as *P*, and the CFD simulation results, denoted as *D* in this paper, is determined as:

$$Error = \left|\frac{(P-D)}{P}\right| 100\% \quad (1)$$

Table 5. Mesh dependence study at Fr 0.25.

Parameter	EXP (P)	Grid 1 (D)	Grid 2 (D)	Grid 3 (D)
Total Resistance (N)	16.46	15.06	15.53	16.64
Error (%)	-	8.51	5.66	1.09

The presented cases demonstrate that the resistance changes monotonically with mesh density, and the comparison shows a reasonably good agreement between simulation (CFD) and experimental (EXP) values. The minimum error can be founded at Fr 0.25, achieved by Grid 3. As a result, Grid 3 is utilized for all case variations, as shown in Table 6.

9. The validation underpredicts the experimental results for Froude numbers in which wavemaking is important. It means that your wavemaking procedure is not converging well enough. Fig 10. also brings up this uncertianty, since you can see the coarse wetted regions. For this reason, explaining point 8 is even more important!
   Answer: We have explained the validation in more detail. We assume the total resistance is the sum of tangential and normal components. In multiphase simulations, RANS solvers compute the total as $C_T = C_P + C_F$, where C_P is the pressure resistance. Shear drag is the tangential vector component of the total surface frictional resistance of the ship to the fluid. Meanwhile, pressure drag is the normal resistance or drag due to pressure which consists of waves and viscous pressure. Pressure and shear forces all components are interrelated and calculating each presents its own challenges, however exceedingly challenging to divide. We also have changed y+ contour in Figure 10.

10. In Table 9 please leave only one decimal, it is too hard to analyze it.
    Answer:
    Thank you for the suggestions. we have used one decimal number.

Table 9. The total resistance of numerical simulation results in all cases..

Fr	*Frh* h_w=0.36 m	*Frh* h_w=2.60 m	Total resistance (*RT*), N							
			Case I	Case II	Case III	Case IV	Case V	Case VI	Case VII	Case VIII
0.25	0.77	0.29	92.2	43.6	11.5	11.3	94.2	36.8	12.5	12.0
0.28	0.87	0.33	126.0	88.0	16.8	16.2	136.4	92.8	17.8	16.9
0.31	0.97	0.36	161.7	113.6	21.7	22.0	183.3	120.9	23.4	22.7
0.33	1.04	0.39	186.2	135.9	26.6	26.7	216.5	150.2	31.2	28.4
0.35	1.1	0.41	209.6	160.3	34.5	33.9	260.4	180.8	39.6	34.2

11. "resistance value increased to 699.69%" is also hard to understand. Compared to what initial value?
    Answer:
    The percentage number is obtained by comparing the RT value in Case I with Case III. The following is a mathematical explanation:

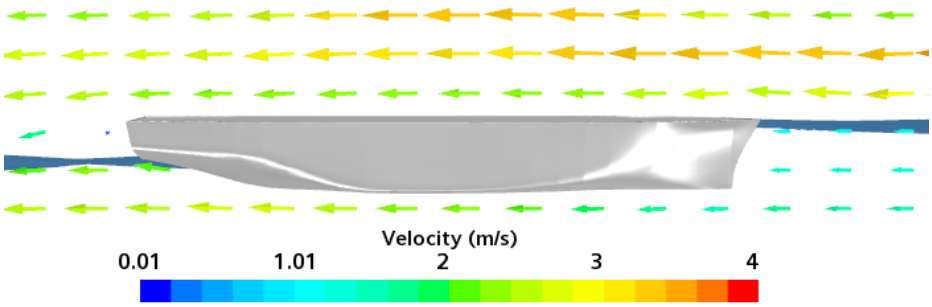
$$\frac{(RT_{Case\ I} - RT_{Case\ III})}{RT_{Case\ III}}\ x\ 100\% = RT\ increase\ Case\ I\ from\ Case\ III\ (\%)$$

    We have updated the texts:
    It can be shown that in Case I, the resistance value increased enormously by about 699.69% compared to the resistance value in Case III. Moreover, Case V also experienced an increased resistance value several times, reaching 654.11% from Case VII.

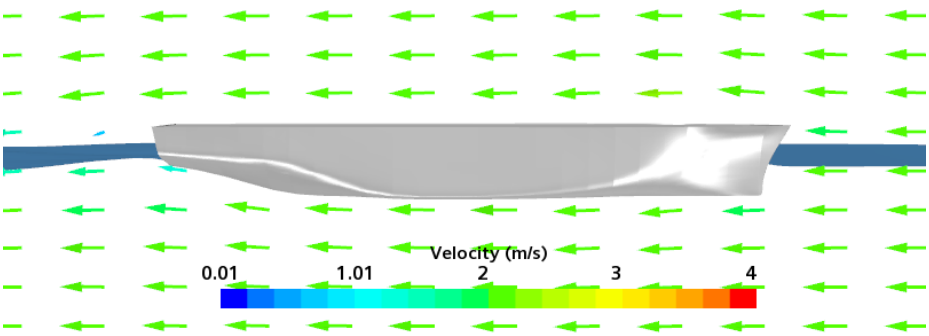
12. The authors should shown how the dynamic pressure field is changing based on the closeness of the bottom in more detail. The image of velocity Fig 20, is not important. Rather, X-Z plane centreplane cut showing dynamic pressure for various depths should be shown. This is the real information on why wavemaking changes, and the resistance force!
    Answer: Thank you for your good idea. The changes are highlighted in blue font colour.

    The pressure distributions for all cases are shown in Fig. 21. The results demonstrate that the pressure distribution around the ship model is more uniform in the case of hw/T = 16.15. However, in the case of hw/T = 2.24, it is seen that a significant high-pressure region is formed at the bow. This observation also explains that pressure increase occurs at low ship-to-bank distances. Relatively heterogeneous pressure contours characterize the pressure variations for narrow and shallow channels. The heterogeneity of the pressure contour is achieved at the bow due to the pressure accumulation of fluid caused by the ship entering a narrow and shallow region. For the keel to stern region, the pressure decreases due to the increase in fluid velocity under the keel. This causes the increase of ship resistance in shallow and narrow channels compared to wide and deep channels, leading to the wave pattern change. Based on these results, the bow is a sensitive area where a significant increase in pressure is identified.

    The flow velocity distribution is shown in Fig. 20



(a)



(b)

Fig. 20 Velocity distribution at Fr 0.35, the blue surface in each plot represents the free water surface : (a) Case V, (b) Case VIII.

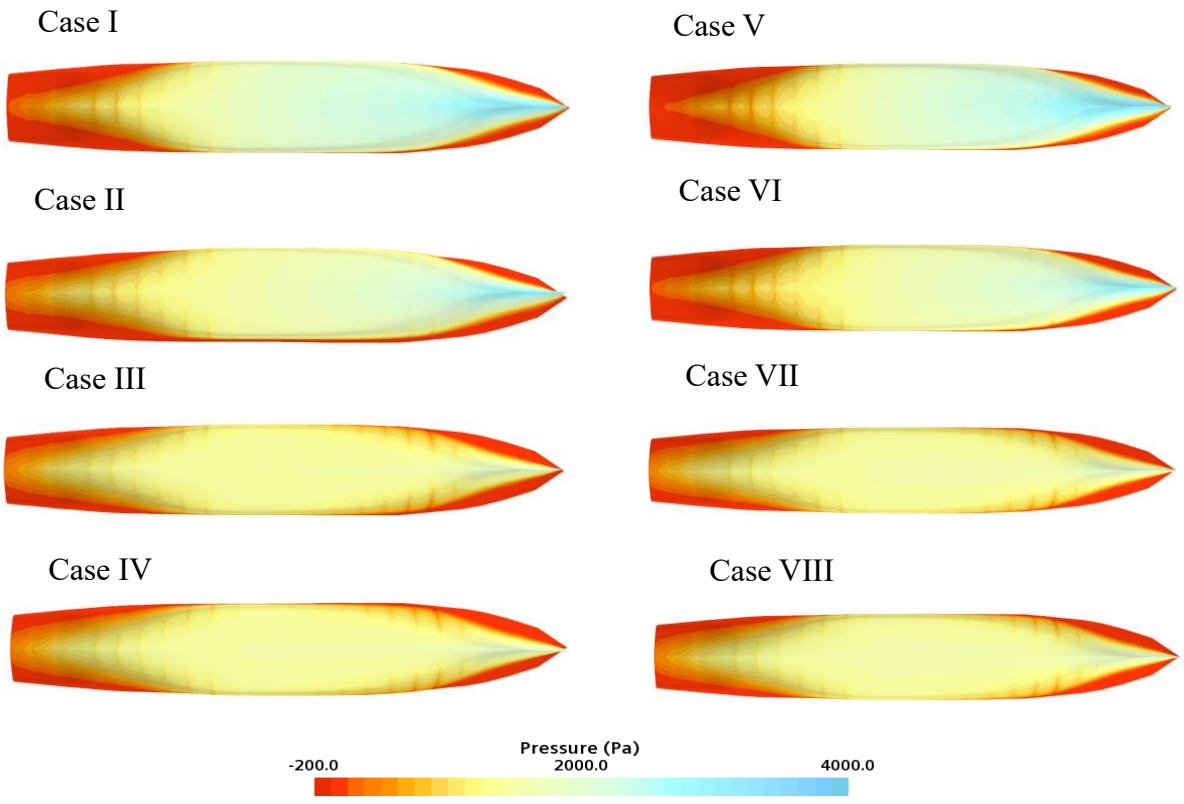


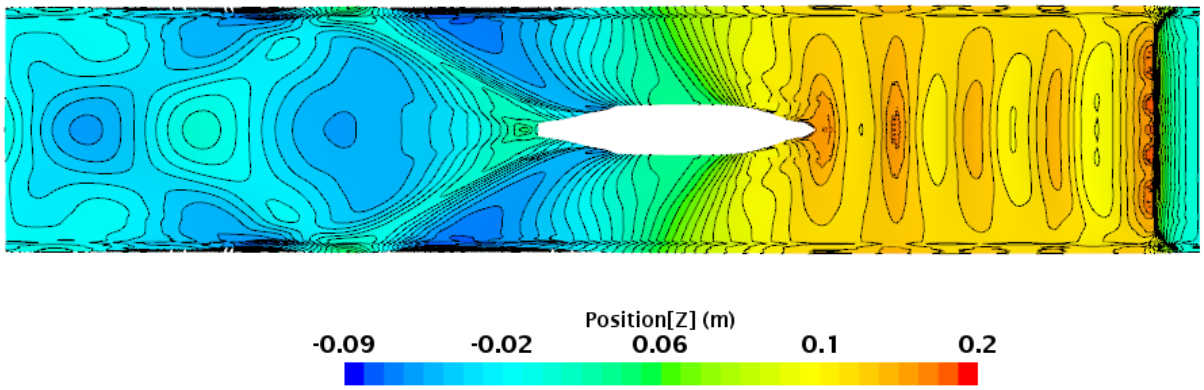
Fig. 21 Pressure distribution in all cases at Fr = 0.35.

13. Please describe physically what happens with help of point 12, as well as your other results, instead of simply comparing the numbers and leaving the rest to readers.
    Answer: we have described physically Figures 20 and 21 in the text.

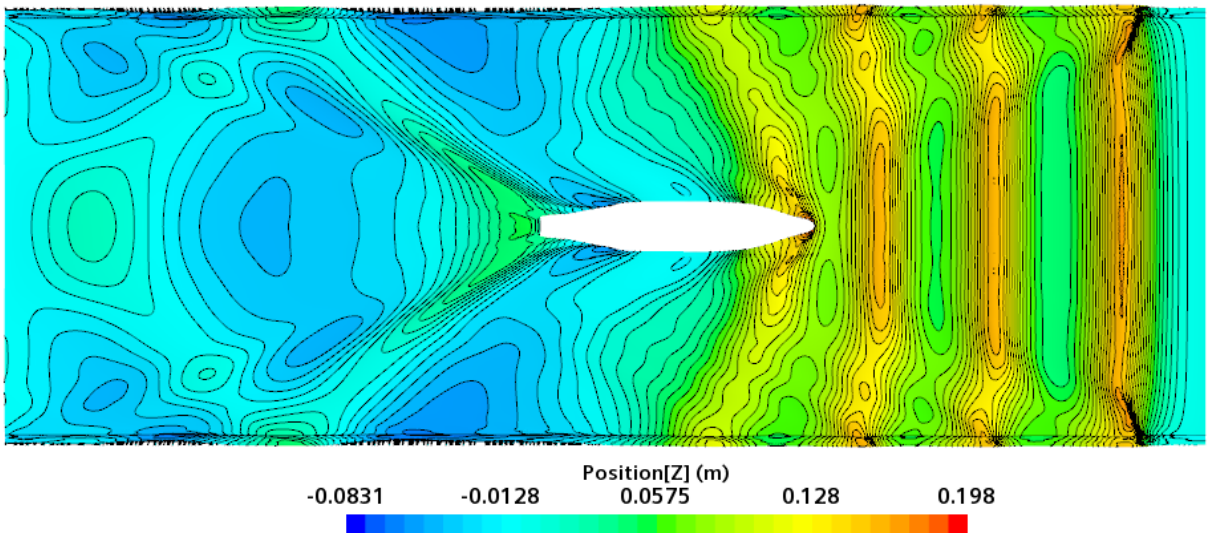
14. Conclusions are wrong/not contributing. After the proper validation and physical analysis, please conclude your findings in a way engineers and researchers can see under which conditions the increase in resistance happens, and physically why.
Answer: We have revised the conclusion as follows:

A series of CFD-based simulations were conducted to investigate the different canal types and cross-sections on resistance and wave-generated characteristics of 750 DWT Perintis Ship. Two different canal types, including canal bank and rectangular canal, were investigated at low and high-speed conditions. In the benchmark study, the proposed CFD simulation results resulted in good agreement with the experimental test, with an error range of 0.11 - 7.74%, indicating the CFD setting and environment can be used in further analysis. The influence of canal types on resistance value results in several phenomena. Similar results were found both in rectangular and canal banks. The case with a shallower (lower h_w/T) and a narrower (lower B_c/B_s) canal dimension has a higher resistance value. The wave pattern generated in the simulation became more complicated as the ship moved through a narrow and shallow canal. Backflow and subsidence of free surface became significant around the ship's hull in more restricted water, changing the ship's hydrodynamic characteristics and increasing resistance. It can be found that the higher the blockage ratio (m_b), the higher the total resistance value in both canal types, which proved that ships with higher speeds were more sensitive to changes in waterway restrictions.
In addition, changes in the dimensions of the canal width at h_w/T = 16.15 have no significant effect on the total resistance. The results demonstrated that the pressure distribution around the ship model was more uniform at h_w/T = 16.15. In contrast, it can significantly affect at h_w/T = 2.24 for both canal bank and rectangular canals. It was seen that a significant high-pressure region was formed at the bow. This observation also explained that pressure increases occurred at low ship-to-bank distances. Relatively heterogeneous pressure contours characterized the pressure variations for narrow and shallow channels. The heterogeneity of the pressure contour was achieved at the bow due to the pressure accumulation of fluid caused by the ship entering a narrow and shallow region.

15. Fig 24. also denotes there may be (surely there is) an issue with overset grid interpolation. please see the corners and comment on this.
Answer: Fig 24 and Fig 25. has been revised as follows :

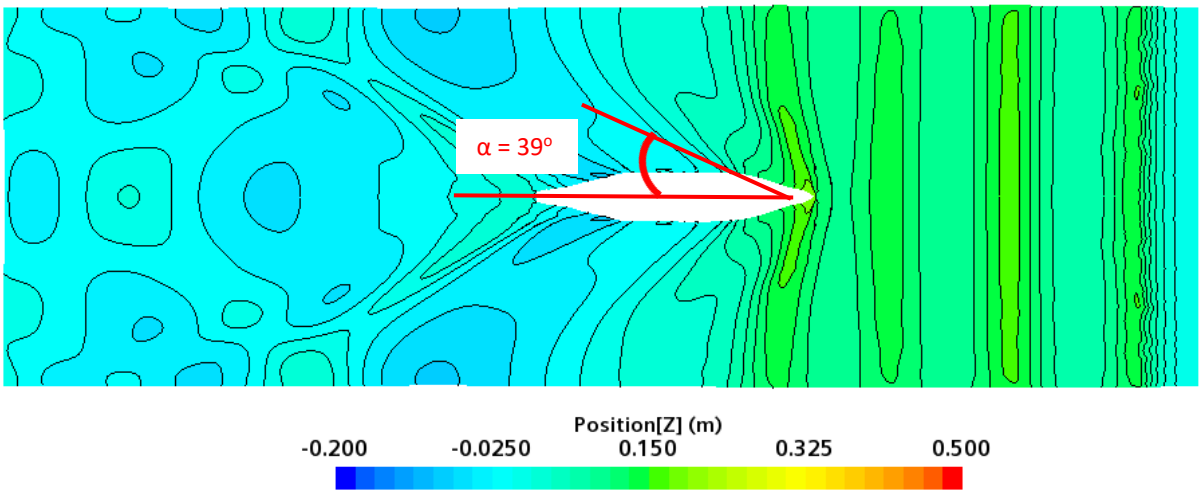


(a)

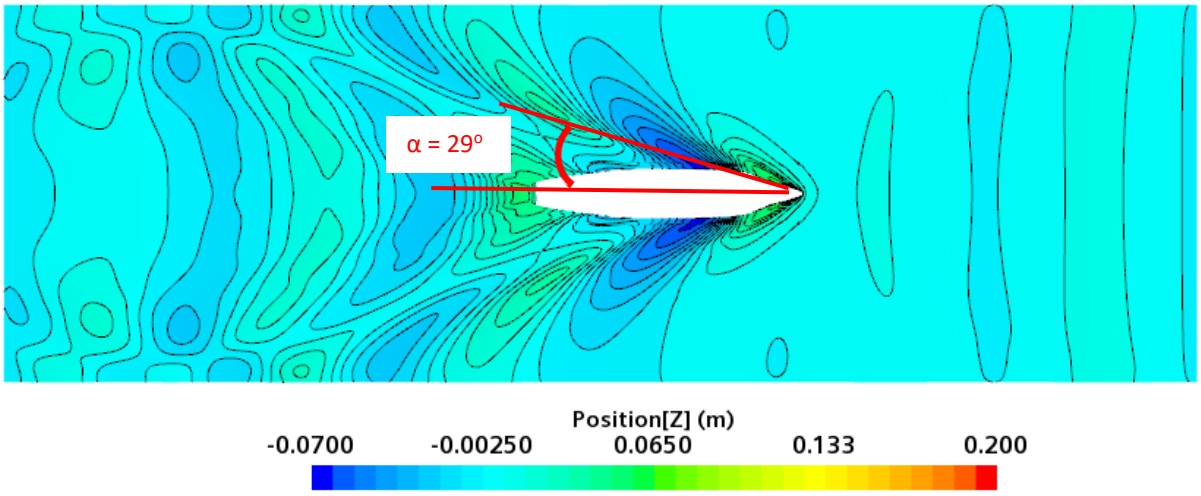


(b)

Fig. 24 Wave pattern at h_w/T = 2.24 and *Fr* 0.35: (a) W_b= 2.4 m (Case I), (b) W_b= 4.8 m (Case II).



(a)



(b)

Fig. 25 Wave pattern at W_b = 4.8 m and Fr = 0.35: (a) h_w/T = 2.24 (Case VI), (b) h_w/T = 16.15 (Case VIII).

Reviewer 2

Thank you for your manuscript. I think this study is worthy of investigation and it is well written in general. However, following points should be revised before the manuscript is considered for publication. Please find my comments and suggestions below. Best regards.

1. Abstract. Following sentence in the abstract does not read well. Please rephrase it. "Although several investigations are found in investigating the vessel behaviour in restricted water, a few research has been carried out in analyzing the different canal types and cross-sections in ship hull form."
   Answer: Thank you for the suggestion. The text has been revised as follows:
   There are many investigations into the behavior of ships in restricted waters, such as ships traveling in different forms of canal cross-sections.

2. Equation (3) Please double check with the governing equation. I think the tensors (i, j, k) are missing.
   Answer: Thank you for your advice. Equation (3) uses nabla symbol (∇), so the tensors (i, j, k) is not written. And the formula has been removed based on consideration of comment number 6 from reviewer 1.

3. Fig. 10. With the current colour bar range, it is not easy to see whether the hull wetted areas have proper wall y+ values (i.e. y+ >30). I suggest the authors either change the upper/lower limits of the colour map, or change the colour map style to solve this issue.
   Answer: Thank you for the suggestion. We have revised wall y+ in the range 0f 0-90. The figure has been revised as follows:

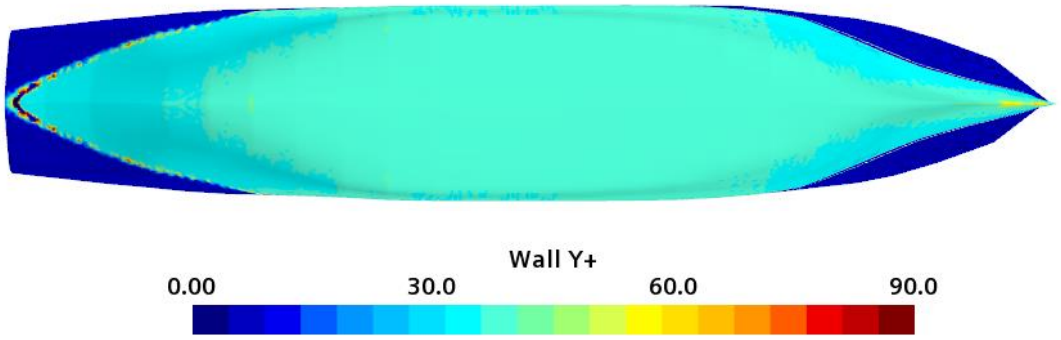


Fig. 10 y+ contour on the Perintis 750 DWT ship hull in the range of (0–90)

4. Boundary condition (Fig. 5-7) I can see that you used Velocity inlet for the bottoms of the tanks, while using no-slip wall for the sides. Do you have any specific reasons? Especially for the canal simulations, I think No-slip wall with velocity (i.e. moving wall) boundary condition might be a better choice. Also, you did not mention whether you applied any velocity for the no-slip wall for the side of the canal. If you did not apply the velocity on the no-slip wall for the sides (to make it a moving wall), the simulation will not represent correctly the flow around the ship and canal walls. To confirm this, can you please show me the velocity contour around the hull including the reason near the side walls?
   Answer: Thanks for the suggestion. We have added the speed contour. The additional figure is as follow:

   Fig. 22 shows that when the ship-bank distance or the water depth reduces, the flow stagnation point on the bow deflects the flow towards the canal wall so that the flow on the side of the ship is significantly accelerated. Further, the flow is directed downward, causing the under-keel flow to accelerate.

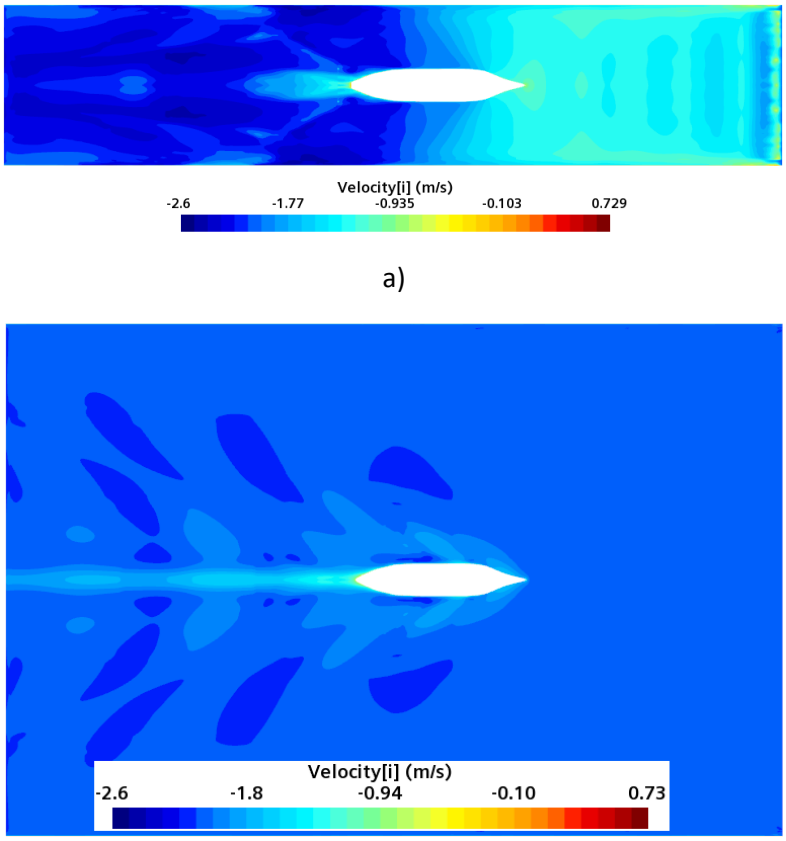


Fig. 22. Water surface x-axial velocity distribution at Fr=0.35 : a) Case I b) Case IV

5. Fig. 13 Please check with the caption of Fig. 13. I think “Cases I-IV” & “canal bank” should be correct.
   Answer: We have corrected the image caption as follows Fig. 13 Resistance value (*RT*) in canal bank (Cases I-IV).

6. Results (Fig. 13-18) Now the results are plotted against the Froude number based on the ship length. I think plotting the results with depth Froude number might be beneficial. Please consider plotting the results with depth Froude number too (maybe you can use 3D figures for both axes, i.e. normal Fr and depth Fr).
   Answer: Thank you for the suggestion. We have decided to compare length Froude number and depth Froude number using two figures. The figure has been revised as follows :

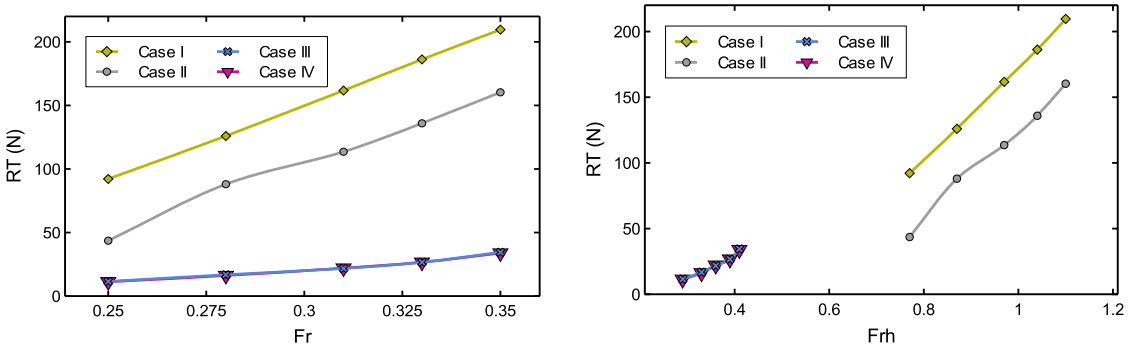


Fig. 13 Resistance value (*RT*) versus Froude Number (*Fr*) and depth Froude Number (*Frh*) in canal bank (Cases I-IV).

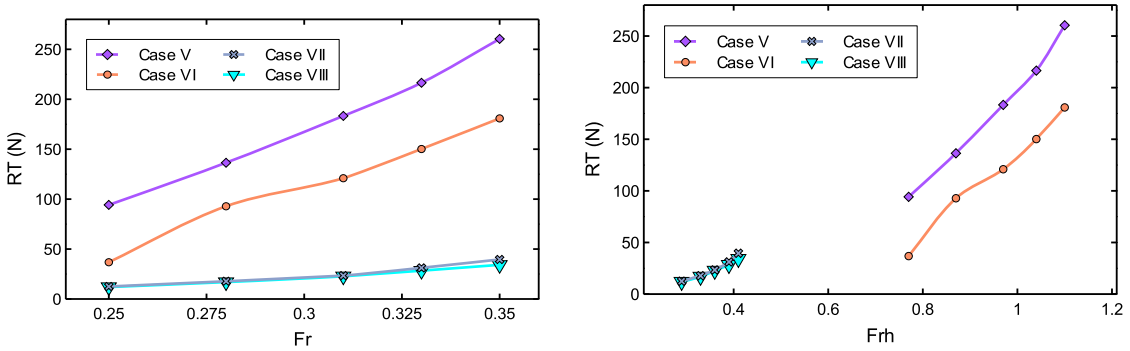
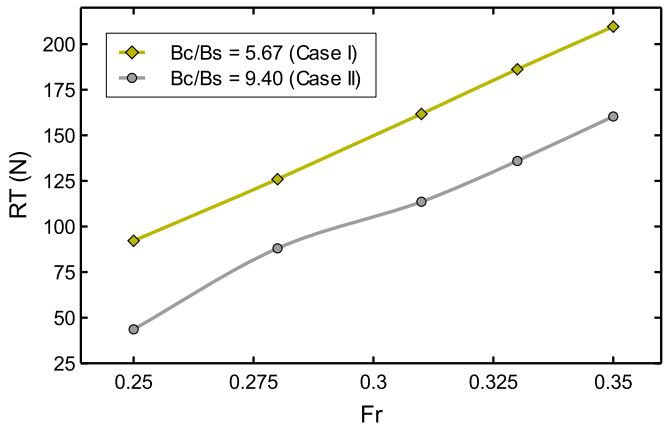
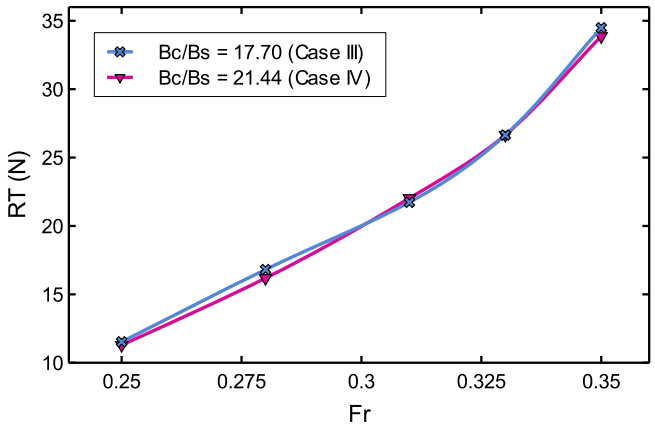


Fig. 14 Resistance value (RT) versus Froude Number (*Fr*) and depth Froude Number (*Frh*) in rectangular canal (Cases V-VIII).

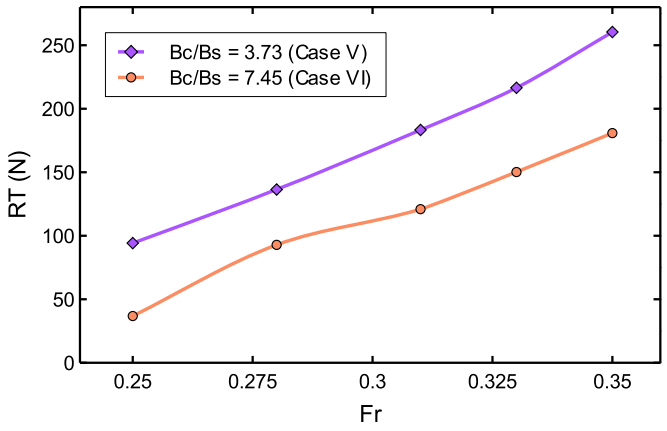


(a)

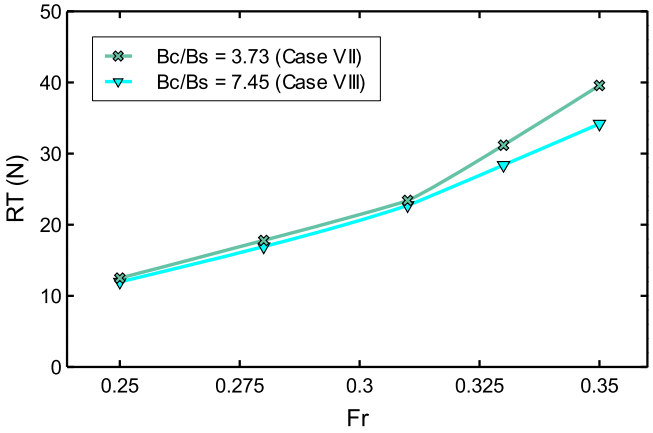


(b)

Fig. 15 Resistance value (*RT*) in canal bank at: (a) h_w/T = 2.24, (b) h_w/T = 16.15.

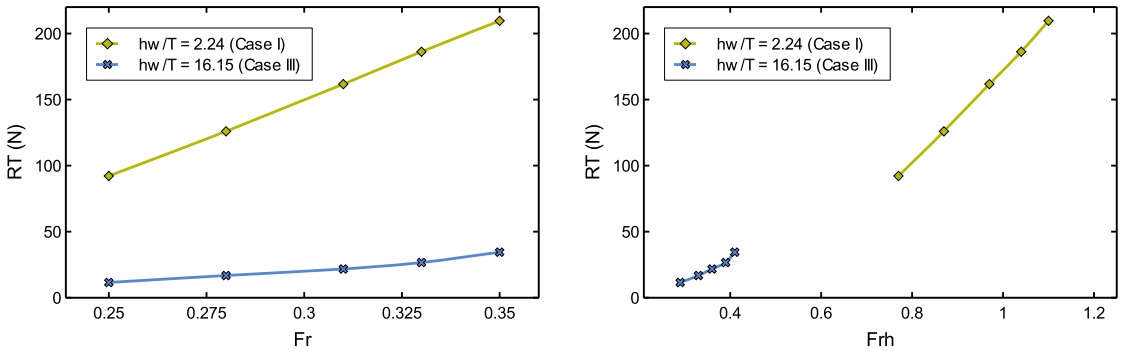


(a)

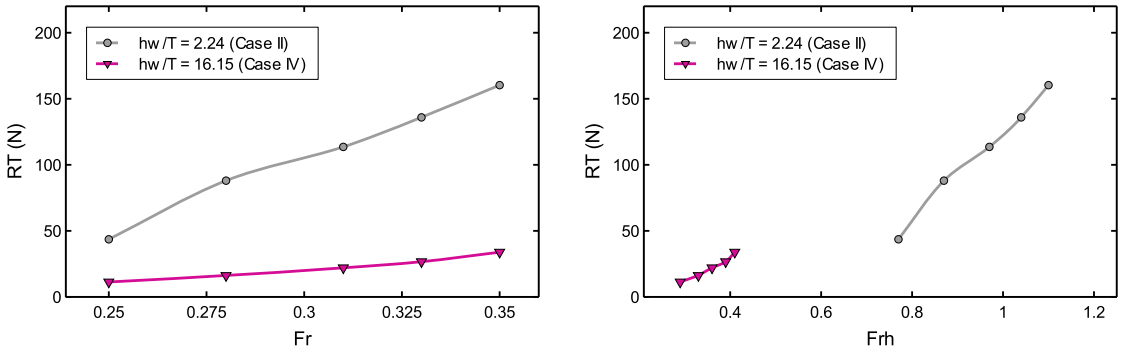


(b)

Fig. 16 Resistance value (*RT*) in the rectangular canal at: (a) h_w/T = 2.24, (b) h_w/T = 16.15.

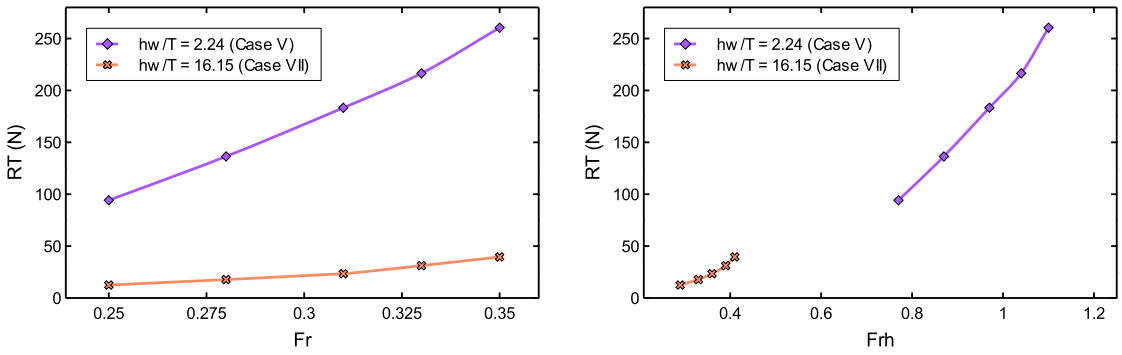


(a)

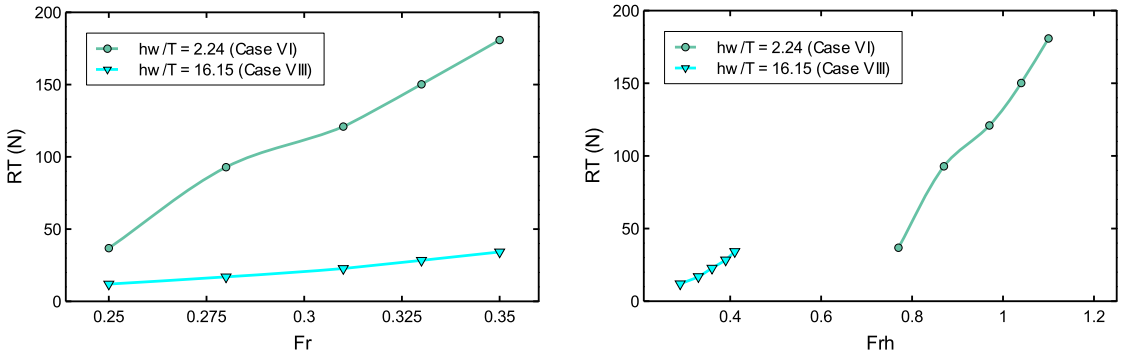


(b)

Fig. 17 Resistance value (*RT*) in canal bank at: (a) W_b= 2.4 m, (b) W_b= 4.8 m.



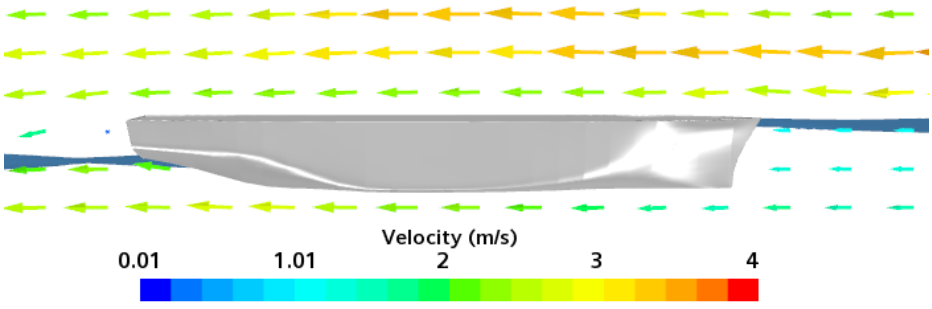
(a)



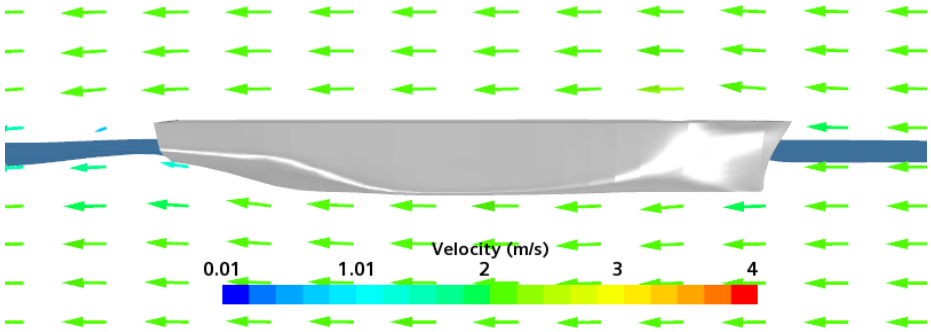
(b)

Fig. 18 Resistance value (*RT*) in the rectangular bank at: (a) W_b= 2.4 m, (b) W_b= 4.8 m.

7. Fig. 20. I am sure that you can change the spacing of the vector arrows. Now the arrows are located at their cell centres, but you can change it to uniform spacing. I am sure that the figure will look much better by doing so.
   Answer: Thank you for the suggestion. The figure has been revised as follows :



(a)



(b)

Fig. 20 Velocity distribution at Fr 0.35, the blue surface in each plot represents the free water surface : (a) Case V, (b) Case VIII.