ITTC2024
30TH INTERNATIONAL TOWING TANK CONFERENCE
22–27 SEPTEMBER 2024 | HOBART TASMANIA AUSTRALIA

Report of the Specialist Committee on Combined CFD/EFD Methods

Specialist COMMITTEE on Combined CFD and EFD Methods

1. INTRODUCTION

1.1. Membership and Meetings

The members of the Specialist Committee on Combined Computational Fluid Dynamics (CFD) and Experimental Fluid Dynamics (EFD) Methods of the 30th ITTC are:

- Prof. Kevin Maki (Chair)
  University of Michigan,
  Ann Arbor, Michigan,
  USA

- Dr. ir. Bram Starke (Secretary)
  MARIN,
  2, Haagsteeg, P.O. Box 28,
  6700 AA Wageningen,
  The Netherlands

- Jussi Martio
  VTT,
  Espoo,
  Finland

- Dr. Joseph Banks
  University of Southampton,
  Southampton
  United Kingdom

- Peter Horn
  Hamburg Ship Model Basin (HSVA),
  Bramfelder Str. 164,
  22305 Hamburg,
  Germany

- Dr. Riccardo Broglia[1]
  CNR-INM,
  Rome,
  Italy

- Prof. Diego Villa
  University of Genoa,
  Genoa,
  Italy

- Dr. Hyunse Yoon[2]
  University of Iowa,
  Iowa City, Iowa,
  USA

- Prof. Feng Zhao,
  China Ship Scientific Research Center (CSSRC),
  Wuxi, Jiangsu,
  China

---

[1] Replaced Stefano Zaghi in October 2022

[2] Originally appointed when working for DSME in Korea, and reappointed when he moved to University of Iowa in February 2023, to fill vacancy left by Marcelo Vitola from LabOceano in Brazil.

- Prof. Yichen Jiang,
  Dalian University of Technology,
  Dalian, Liaoning,
  China

- Dr. Eng. Shoji Shingo
  Shipbuilding Research Centre of Japan (SRC),
  Tokyo,
  Japan

- Dr. Stefano Zaghi[3]
  CNR-INM,
  Rome,
  Italy

- Marcelo Vitola[4]
  LabOceano,
  Brazil

Three in-person committee meetings have been held during the work period:

- The first was held in Ann Arbor, MI, May 10-12, 2022. The venue was the Michigan League, on the campus of the University of Michigan. There were six in-person participants, and four members joined the meeting virtually.

- The second meeting was held in Hamburg, Germany, April 18-20, 2023. The meeting was hosted at HSVA. There were 10 in-person participants, and one joined virtually.

- The third meeting was held in Tokyo, Japan, May 8-10, 2024. The meeting was hosted by SRC. There were ten committee in-person members, two in-person guests, and one virtual participant.

- The committee met many times virtually to prepare reports, prepare for the in-person meetings, and to discuss the progress of the work of the committee. There were approximately six virtual meetings per year.

2. TASKS

The recommendations for the work of the Specialist Committee on Combined CFD and EFD Methods as given by the 29th ITTC were as follows:

---

[3] Left his institute in May 2022, and resigned from committee. Was replaced by Riccardo Broglia.

[4] Was originally appointed to committee but left his institute before committee activities started. His vacancy was filled when Hyunse Yoon moved from Korea to USA.

1. Review and highlight good examples of combined methods, suggest and initiate new applications of combined methods, and co-ordinate and encourage each technical committee to perform detailed work on combined methods.

2. Co-ordinate and advise each technical committee to investigate and develop combined methods.

3. Monitor and review advances and challenges within full-scale and model scale CFD with special focus on speed/power predictions.

4. Review the outcome of ongoing CFD benchmark campaigns.

5. Encourage the establishment of open validation data for high Reynolds number flow cases for marine applications.

6. Review and study the performance of turbulence models and wall treatments at full scale. Monitor the development of new turbulence modelling approaches when they become available.

7. Monitor advances in the application of detailed flow measurements in the ITTC community.

8. Develop a standard process of performing a CFD benchmark study within ITTC.

9. Monitor how Verification and Validation is applied and reported in research publications and commercial work.

10. Continue to maintain and improve the existing Recommended Procedure 7.5-03-01-01, "Uncertainty Analysis in CFD, Verification and Validation Methodology and Procedures".

11. Monitor the use of the new Recommended Procedures 7.5-03-01-02 "Quality Assurance in CFD Ship Applications" and update it if needed.

12. Produce information material (articles, conferences, social media) directed towards stakeholders who receive and use the results hydrodynamic predictions. Explain state-of-the-art capability, and challenges of CFD versus EFD and Combined methods.

13. Update, as required, procedure 7.5-03-01-01, Uncertainty Analysis in CFD, Verification and Validation Methodology, and Procedure 7.5-03-01-02, Quality Assurance in CFD Ship Applications.

3. LIST OF ABBREVIATIONS:

ABL	Atmospheric Boundary Layer
AIAA	American Institute of Aeronautics and Astronautics
ASME	The American Society of Mechanical Engineers
AVT	Applied Vehicle Technology
BPG	Best Practical Guideline
CAD	Computer Aided Design
CF	Correction Factor
CSSRC	China Ship Scientific Research Center
DES	Detached Eddy Simulation
DIC	Digital Image Correlation
EEDI	Energy Efficiency Design Index
EEXI	Energy Efficiency Existing Ship Index
ESD	Energy Saving Device
FS	Factor of Safety
GCI	Grid Convergence Index
HVAF	Hub Vortex Absorbed Fins
HPC	High Performance Computing
IACS	Safer and Cleaner Shipping
IDDES	Improved Delayed Detached Eddy Simulation
JBC	Japanese Bulk Carrier
KCS	KRISO Container Ship
KRISO	Korea Research Institute of Ships & Ocean Engineering

LES	Large Eddy Simulation
LSR	Least Square Root
NDA	Non-Disclosure Agreement
PIV	Particle Image Velocimetry
PPTC	Potsdam Propeller Test Case
PSD	Pre-Swirl Duct
RANS	Reynolds-Averaged Navier-Stokes
Re	Reynolds number
RE	Richardson Extrapolation
RP	Recommended Procedures
SC	Specialist Committee
SRC	Shipbuilding Research Centre of Japan
STO	Science and Technology Organization
SVA	Schiffbau-Versuchsanstalt Potsdam
TKE	Turbulent Kinetic Energy
V&V	Verification and Validation
VVUA	Verification, Validation, and Uncertainty Assessment

4. UPDATES OF THE RECOMMENDED PROCEDURES

4.1. Background

Validation and Verification **(V&V)** in the field of maritime numerical fluid dynamics is a basic tool to assess the uncertainty of a calculation or simulation as well as to guarantee the accuracy and reliability of a result. There are generally two approaches to address these aspects described in two different ITTC procedures 7.5-03-01-01, “Uncertainty Analysis in CFD, Verification and Validation Methodology and Procedures” and 7.5-03-01-02 “Quality Assurance in CFD Ship Applications”. The first one aims for a detailed mathematical look into the calculation or simulation itself, whereas the latter one addresses the accuracy of performed CFD calculations based on best practice guidelines on a statistical view.

TOR 9 addresses the application of the first procedure in commercial and academic work. In close connection to this, TOR 10 deals with the modification of this procedure. The usage of the second mentioned procedure is subject to TOR 11. Finally, TOR 13 requests the specialist group to update both procedures.

Focusing on the application of both procedures within commercial and scientific work, the ITTC specialist group has set up a survey among ITTC members to see if this procedure is used in their work. The following section represents the results of this survey, preceded by two sections of details on the update of both these mentioned procedures.

Following this, a final section summarizes the review and update of further ITTC guidelines and procedures reviewed by this committee.

4.2. Survey on V&V and Quality Assurance in CFD

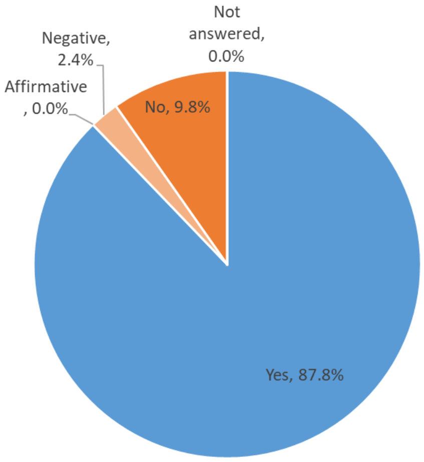
The present committee prepared and distributed a questionnaire to ITTC members to share their vivid experiences and useful comments. The questionnaire was distributed to 102 member organizations, of which 42 organizations responded. These organizations comprised 20 research institutes, 13 universities, and 9 commercial companies. The regional composition is 4 Americas, 6 Central Europe, 5 North and West Asia, 5 Northern Europe, 11 Pacific Islands, 7 Southern Europe, and 4 South and East Asia of ITTC Geographical Area countries. Of these, two organizations submitted two separate responses, resulting in a total of 44 responses. Of these, 3 responses were deemed invalid, thus finally 41 valid responses were collected.

The survey is a two-part questionnaire consisting of Part A and Part B. Part A contains 11 questions (A1 ~ A11), and Part B contains 5 questions (B1 ~ B5). Questions A1 through A5 are related to TOR 9, A6 through A11 to TOR 10, and B1 through B5 to TOR 11. Here, each question statement is cited in italics and the responses are presented using a pie chart for each question. In the case of YES or NO response format, if the answer was non-explicit but the meaning was positive, it was classified as ‘Affirmative,’ whereas, it was classified as ‘Negative’ if the meaning was disagreeing. For each question, the responses were briefly discussed and the narrative responses, if any, were summarized. Finally, an overall summary and concluding remarks are given at the end.

Part A

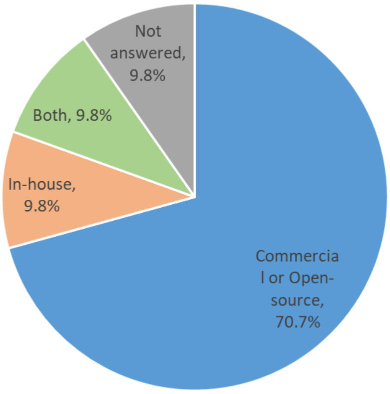
The following questions (A1 – A5) are about CFD Verification and Validation in general.

A1. *Did you ever prepare a V&V study for CFD simulation? If your answer to A1 is YES, how often do you prepare V&V studies and what kind of V&V methodology do you use? If possible, please let us know relevant reference(s) to the methodology. If your answer to A1 is NO, please let us know the reason (then, you may skip A2 through A5 and continue from there).*



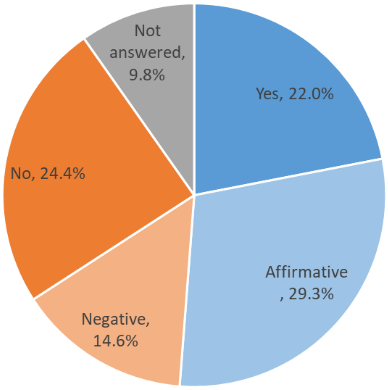
A vast majority (>85%) has ever carried out or have been involved in V&V studies and are mainly using the well-known procedures and papers which are relevant within this context (papers by the main authors, for example, Richardson (1911), Roache (1994), Eça and Hoekstra (2014), Stern et al. (2001), next to ITTC (2021), The American Society of Mechanical Engineers (ASME) (2009), American Institute of Aeronautics and Astronautics (AIAA) (1995) procedures). Those who replied “No” can be considered as pure experimental facilities or institutes who are planning to do V&V in near future.

A2. *When you perform V&V studies, do you make it for a commercial CFD code, or do you use an in-house developed CFD code?*



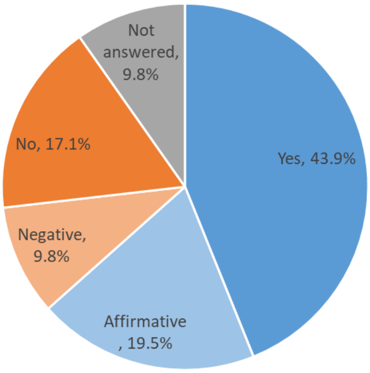
Commercial or Open-Source Codes are widely spread among the participants of this survey. About 70% of the respondents are using either Commercial or Open-Source Codes for V&V studies. Institutes only using in-house developed codes are 10% of the replies and another 10% of the replies using both In-house and Commercial/Open-source codes for their V&V studies. About 10% of the replies are without an answer. Specific CFD codes mentioned in the survey responses include OpenFOAM, STAR-CCM+, FINE/Marine, FINFLO, NEPTUNE, SURF, NAGISA, CFX, ReFRESCO, FreSCo+ and Fluent.

A3. *Do you prepare V&V studies for CFD projects which can be considered as daily business in your organization?*



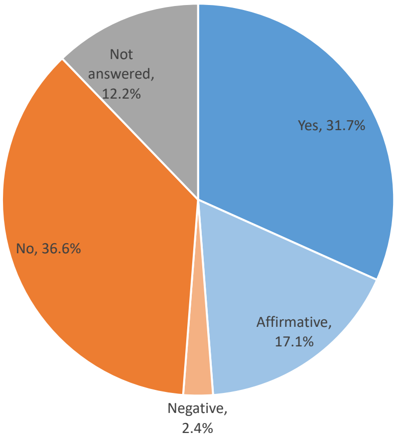
Half of the respondents replied “Yes”, and the other half replied “No”. Some rely on their developed best practices which are based on investigations which include V&V studies, so a V&V study is not necessary for everyday projects. Others do such a study for all daily projects although it is very time-consuming. Of course, the extent of such a V&V study can be very different. This result gives a good indication of the awareness for a V&V study: either they are done for a project, or the calculations are based on the settings verified with earlier V&V studies.

A4. Do you prepare V&V for new types of simulation cases or new ship types?



A majority (~65%) replied yes or gave a rather positive answer. Preparation of a new V&V study is then carried out when new ship types or simulation types must be calculated. It depends rather on the difference and the novelty of the new calculation case if a complete V&V study is carried out or not. So, there is no clear differentiation when a participant defines a new simulation as “new enough” or “very different to what was done before” that it is decided to carry out a complete V&V study.

A5. Do you perform Energy Efficiency Design Index (EEDI) / Energy Efficiency Existing Ship Index (EEXI) relevant CFD simulation in your organization and prepare a V&V study for this work as well?

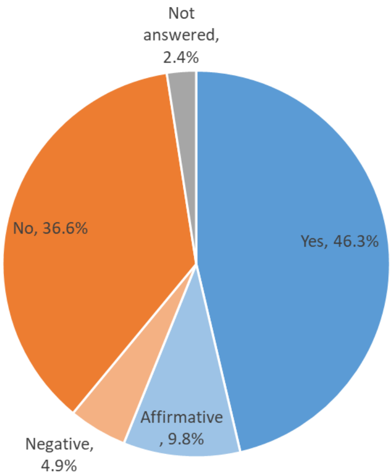


Unfortunately, the question is not posed perfectly as it covers two questions in one. The results cannot be used to distinguish between an answer “No, we do not carry out EEDI/EEXI calculation” and “We do not make a V&V Study for EEDI/EEXI calculation. Therefore, an evaluation can led to wrong conclusions.

Nonetheless, about 32% answered “Yes” indicating that they carry out CFD simulations for EEDI/EEXI and perform V&V studies for this. Some answered that they were going to prepare V&V for EEDI/EEXI in the future.

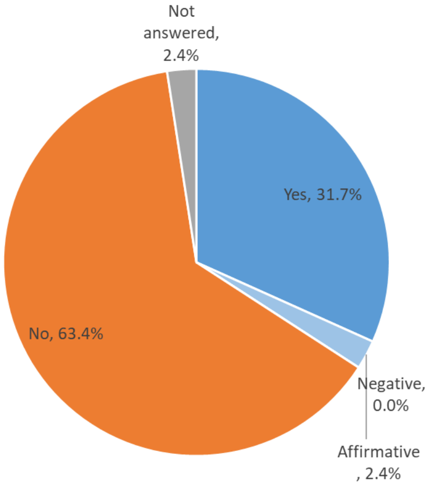
The following questions (A6 – A11) are about RP 7.5-03-01-01 “Uncertainty Analysis in CFD Verification and Validation, Methodology and Procedures.”

A6. Do you use this procedure for your organization’s activities?



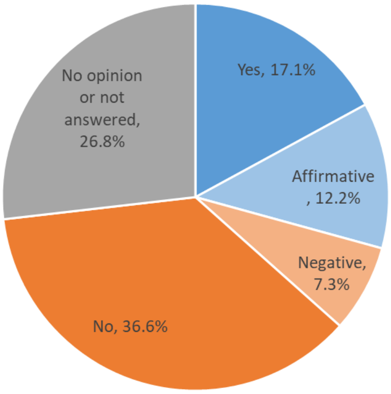
About 46% said they use this procedure, while about 37% said they do not. About 10% said they use it partially, and about 5% said they rarely use it or do not apply it to their daily tasks. In cases where they were partially used, they responded that they used definitions that were somewhat different from this procedure or that they mainly used ASME's procedures (ASME, 2009).

A7. Have you participated in benchmark studies where you have to prepare a verification study following this procedure? If your answer to A7 is YES, but you did not (or were not able to) submit your verification results, may we know what caused this?



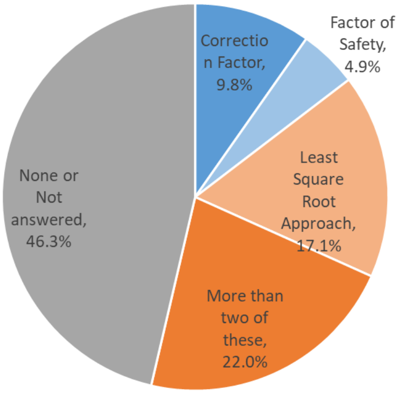
Approximately 32% responded that they had experience participating in benchmark studies requiring V&V, while 63% responded that they had no such experience. One respondent said they had not had this experience recently. Three respondents mentioned that they had participated in the Gothenburg 2010 Workshop (Larsson, 2014) and conducted this benchmark study. One respondent noted that they had participated in this benchmark study, but that creating similarity grids was very time-consuming and that a grid sensitivity study would have been more practical.

A8. Verification studies following the Richardson Extrapolation may not tend to be asymptotic in general. Do you think that other approaches are more suitable? If your answer to A8 is YES, please let us know other approaches that you think more suitable.



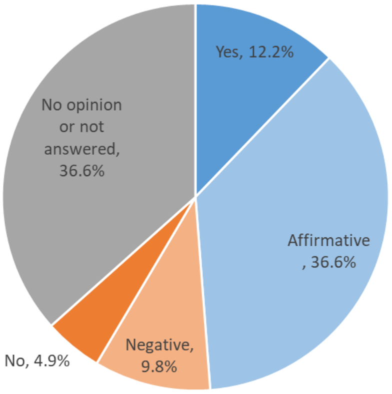
Approximately 17% responded that other approaches were more appropriate than the Richardson Extrapolation (RE) approach (Richardson, 1911), and another 12% agreed with this opinion. Many respondents expressed a preference for using the Least Square Root (LSR) method (Eça and Hoekstra, 2014), mentioning its advantages such as ease of application to unstructured grid systems, unsystematic grid refinement, and oscillatory convergence cases. Nevertheless, some respondents pointed out that the need for more than four grids is a disadvantage of the LSR method. Meanwhile, about 37% of respondents did not agree that a more appropriate approach is needed than the RE method, and another 7% answered that although the RE approach may have drawbacks but is still reliable. What is noteworthy is that about 27% of respondents had no opinion or not answered, citing reasons such as lack of knowledge or unfamiliarity with other possible approaches or methods that could replace the RE method.

A9. Do you follow the concept of the "Correction Factor" (item 4.3), the concept of "Factor of Safety" (item 4.4) or the concept of "Least Square Root Approach" (item 4.5) when you perform a verification analysis? Can you give an explanation?



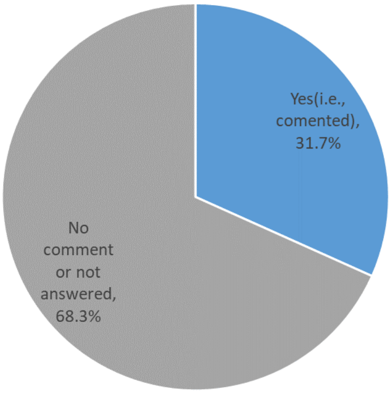
About 10% of respondents said they were using the Correction Factor (CF) approach, about 5% were using the Factor of Safety (FS) approach, and about 17% were using the LSR approach. Another 22% reported using two or more different methods. Among respondents who said they used CF or FS methods; some cited the simplicity and quick application of both methods as a reason for their choice. For respondents who answered that they use the LSR approach, the advantages mentioned in the previous question (i.e., easier application to unstructured grid systems, unsystematic grid refinement, and oscillatory convergence) were explained as reasons for choosing this method. Among the respondents who answered that they use two or more methods, some added that they apply the CF or FS method first because these are faster, and that they apply the LSR method if the condition of monotonic convergence required for RE is not met. Of the remaining 46% of respondents, about half said they did not use all three methods presented, and the other half did not respond.

A10. Do you see room for improvement of this procedure? Do you see this procedure as practical and applicable?



Approximately 12% said that the procedure needs improvement, and another 37% agreed. Most respondents said they believed the process was practical and applicable, but that there was still room for improvement. One of the most common improvements mentioned is the addition of as many examples as possible, allowing users to follow the procedures step by step. Respondents indicated that current procedures are generally described as too difficult to understand and therefore generally difficult to apply. Other respondents answered that the use of this procedure requires the creation and calculation of too many grid cases and that it is difficult to apply in practice even for systematic grid refinement.

A11. Do you have general remarks on the procedures of Validation and Verification studies and future fields of research?

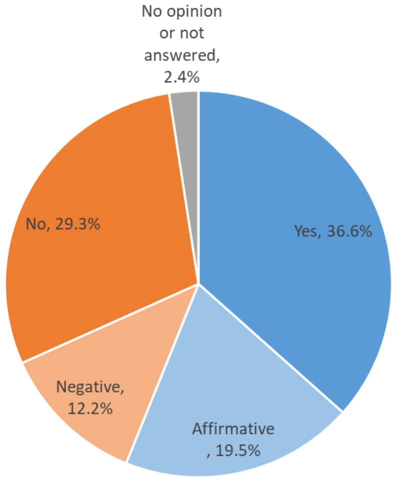


Approximately 32% left comments regarding this process and future improvements. While some say that this procedure was well designed and developed faithfully to the basic assumptions, there were several critical opinions, such as the ones below. While this procedure emphasizes grid studies, it has been suggested that the uncertainty analysis should include a variety of other error factors that affect the solution, such as resolution within the boundary layer and near-wall spacing. There was also criticism about the strict convergence conditions of the numerical solution that did not reflect reality. For example, even in cases where numerical solutions on different grids appear to be diverging or where large uncertainty is predicted, the difference between the coarsest grid and the finest grid solution can be less than 1-2%. Additionally, there are concerns that the systematic grid refinement process may have an undesirable effect on the convergence of numerical solutions. For example, when wall functions are used for turbulence prediction, they point out that wall boundary conditions can be affected if grid refinement is not done carefully enough. Although there are procedures that can improve the case of grid refinement involving the wall function, they may not be practical for typical applications. There is also criticism that this procedure is quite limited in its mention of uncertainty due to time discretization and that there is no recommendation for the prediction procedure. When the error or inaccuracy prediction procedure presented in this procedure is applied to a point variable, the convergence of the numerical solution is often not satisfied, and some argue that an integral-based procedure should be developed to prepare for such cases. Lastly, as an effort to improve the V&V procedures required for EEXI, which is a recent issue, some organizations expressed their willingness to participate in case a benchmark study was needed.

Part B

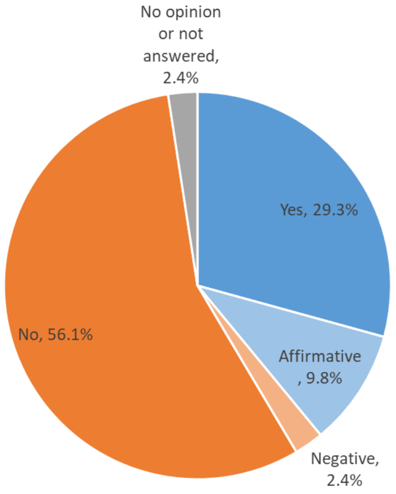
The following questions (B1 – B5) are about Quality Assurance in CFD Ship Applications

B1. Do you use RP7.5-03-01-02" Quality Assurance in CFD Ship Applications" for your organization's activities?



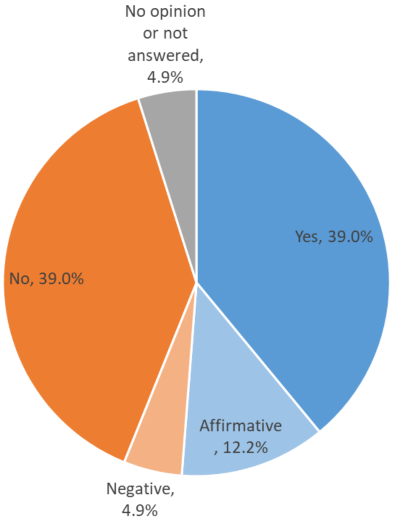
A majority (>63%) replied yes or affirmative answer. Here, affirmative comments are such as “using similar but not identical one” and “planning to use it.” Negative comments are “will review later” and “now using the previous RP (2017 edition)". There were no comments from respondents who answered “No”. So, it’s unclear why they don’t apply Quality Assurance in CFD in their activity. In any case, 36% of the respondents are using RP7.5-03-01-02 “Quality Assurance in CFD Ship Applications” itself and some organizations are using similar ones. It was confirmed that Quality Assurance in CFD is put to practical use.

B2. Do you use RP7.5-03-01-02" Quality Assurance in CFD Ship Applications" for your organization's activities? If your answer to B2 is YES, please let us know in what field did you use the RP, and to what kind of organization did you provide the results. If your answer to B2 is NO, could you please motivate why not?



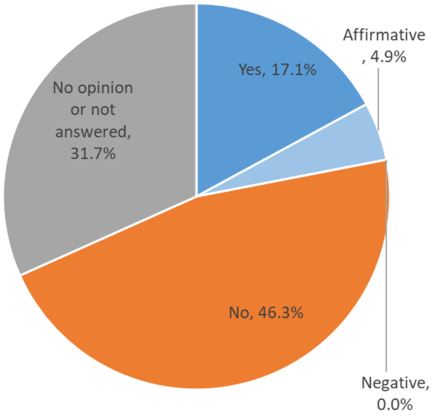
Procedures based on the Best Practical Guideline (BPG) were just included in this RP at the last conference, so it was found that the number of organizations strictly applying it was relatively small (34% of respondents). Despite the above, various cases of application were reported as follows. Application areas: Ship propulsive performance including EEXI related project and ship trim optimization, Ocean structure dynamics, Wake field, Propeller open water characteristics, Wind load for the superstructures. Providing to: Ship-owner, Design office, Class society, Academic application, Organization about ship hydrodynamic conference. Comments from respondents who answered "No" were such as "no opportunity", "not any serious problems", "Not required", "No needs from customers" and "Not necessary for educational purpose". Others commented that they are using similar or almost the same guidelines.

B3. Are you planning to provide CFD and EFD results using the Best Practical Guideline in accordance with RP7.5-03-01-02 in the near future? If your answer to B3 is YES, please let us know in what field you will use the RP and to what kind of organization you will provide the results.



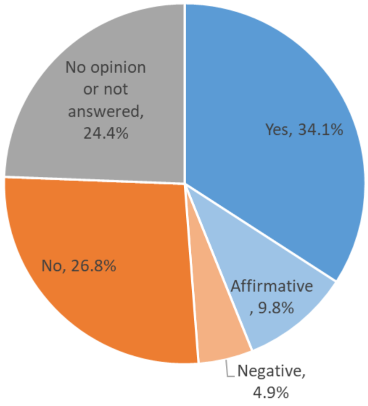
Related to question B.2, B.3 is the question regarding the future application of BPG. There are some relatively affirmative comments from several organizations such as "probably", "maybe" and "if required or mandatory". As a result, about half of the answers were yes or affirmative. Application areas, in addition to the answers in B.2, are: Nautical bottom effect, Ship seakeeping performance and Ship manoeuvrability performance. Shipbuilders will be added to the providing destination in the future. This indicates the utilize of BPG will expand more widely.

B4. Do you think there are any deficiencies in RP7.5-03-01-02? If your answer to B4 is YES, please describe them.



A vast majority (78%) has no opinion or opinion that there is no deficiency in this RP. On the other hand, a few organizations found deficiencies and pointed them out as follows. It is limited to steady simulations and its applications to unsteady/dynamic simulations are not clear. Following this RP would require a lot of computer resources. The current guideline is too concise and needs more detailed descriptions (Various turbulence models were introduced and some viewpoints for grid quality evaluation were described).

B5. Do you think there are any deficiencies in RP7.5-03-01-02? Do you need more detailed, standardized guidelines on how to present statistics for results preparation and delivery?



44% of the respondents requested that the guidelines should be improved in some way, and more than half of the respondents answered “No” or “No opinion”. Some of the comments, regardless of the respondent’s position, are as follows. We should try to concentrate on a handful of universal significant statistical parameters to concentrate/qualify as much as possible the reference standards. Not everyone has access to large databases of trials data, so statistics can be difficult to obtain. We hope to receive an additional description regarding sample size. The guideline has just been proposed and should be maintained for a while to gather information of the applicants. It would be better to provide more examples in the guidelines. It is important that the significance and effectiveness of V&V be widely recognized. In addition, it was pointed out that there are discrepancies in V&V studies due to selectable uncertainty analysis methods, and misunderstandings regarding the uncertainty of numerical simulations among non-experts.

Summary and Concluding Remarks. The organizations and institutes that responded to the questions are generally aware of the importance of conducting validation and verification studies for maritime applications. There is clearly sufficient experience in conducting V&V, as a large majority responded that they have already conducted such a study. It makes no difference whether a commercial or an open source CFD code is used: V&V studies are conducted for both types of flow solvers. It can be concluded from this that around half of the respondents see carrying out such studies as their day-to-day business, if the time permits. On the other hand, around two-fifths of the respondents are not able to carry out V&V studies on a day-to-day basis. It can be pointed out that a validation study is not carried out for all simulation tasks, but for new types of ships or calculations. This suggests that the distinction between when a simulation task is considered to be a new type of calculation or not (or whether a new type of ship is in focus) is not clearly defined and is left to the institute or organization to decide.

Most of the respondents indicated that they had experiences performing V&V for CFD, however, it is not encouraging to see that only slightly more than half said they were using RP 7.5-03-01-01. Additionally, the fact that only about one-third of the respondents had experience participating in a benchmark study that required V&V results using this procedure may be evidence that this procedure may not be used as actively as desired in the CFD community. Looking at the respondents' answers, one of the reasons is that the current procedure may be described as too difficult to be easily applied to real problems, and there are insufficient step-by-step examples to facilitate understanding. Among the three V&V approaches provided in this procedure, many respondents use the LSR method, which they say has the advantage of being relatively easy to apply to unstructured grids, less stringent on the requirements of systematic grid refinement, and practical application. It is also applicable to cases of oscillatory convergence that often occur. However, respondents also pointed out that the too large number of grids required for V&V approaches are inefficient in terms of time and computational cost for application to real-world problems, so some respondents preferred using CF or FS methods as these methods are faster in calculation and require fewer number of grids. Many respondents agreed that this process needs improvement, anticipating the above-mentioned difficulties resolved.

Through this survey, we could monitor the current use of the new RP 7.5-04-01-02 “Quality Assurance in CFD Ship Applications” accurately, and the future use of it was also found. More than half respondents are already using the RP 7.5-04-01-02 itself or something similar. It was confirmed that this RP and Quality Assurance in CFD are currently in practical use. The BPG is used by a small number of organizations, but various applications of it were reported, and the expansion of the use of BPG can be supposed from the comments of the questionnaire. Concerning deficiency in the RP 7.5-04-01-02, The vast majority either think that it has no deficiency or have no opinion about this. On the other hand, a few organizations found deficiencies in it and gave us their opinions. Those opinions will be helpful when this RP is revised.

4.3. Update of RP 7.5-03-01-01

RP 7.5-03-01-01 is updated based on the results of the survey discussed in the previous section. The introduced updates first limited the use of maybe too detailed formulas for deriving the variables related to uncertainty analysis as much as possible. Instead, the updated guide provides references to several published methods useful to conduct Verification, Validation, and Uncertainty Assessment (VVUA) as it relates to CFD for the ITTC, together with a current set of definitions of the different aspects of the field of VVUA.

The updated guide down-selects two different V&V methods, instead of the three methods provided in the old guide, which are the Correction Factor (CF) method, Factor of Safety (FS) method, and the Least-Square-Root (LSR) method. The CF and FS methods are replaced with the method of Stern et al. (Xing and Stern, 2010). The LSR method is retained but referred as the method by Eça and Hoekstra (Eça and Hoekstra, 2014). Step-by-step examples are provided by applying the two methods to practical problems, reflecting the demand raised from the survey outcomes discussed in the previous section.

This section first presents brief outlines of the two methods. Next, the two methods are compared to each other by using examples applying them to practical problems to show how both methods are equally suitable and the differences between them, with some aspects that users are to be careful about while using them. Lastly, future works are discussed.

Common Background. Both methods are based on the generalized Richardson Extrapolation (RE) and use an asymptotic expansion of the numerical solution S that depends on the step size h as

$$S = S_0 + \alpha h^p + o(h^p) \qquad (1)$$

where, S_0 is the exact solution, α is an unknown constant, and p is the order of accuracy of the simulation. Then, the numerical discretization error is estimated as

$$\delta_{RE} = S - S_0 \qquad (2)$$

The Method by Stern et al. This method solves Eq. (1) for the unknown p, with ignoring the higher order terms $o(h^p)$, by using a set of three different grids (a grid-triplet) that are refined systematically. For solutions, say, S_i, with i = 1, 2, and 3 representing the find, medium, and coarse grid, respectively, p is found to be

$$p = \frac{\ln(\varepsilon_{32}/\varepsilon_{21})}{\ln(r)} \tag{3}$$

where, $\varepsilon_{21} = S_2 - S_1$ and $\varepsilon_{32} = S_3 - S_2$ are the differences between different solutions, and $r = h_2/h_1 = h_3/h_1$ is the ratio of the systematic grid-refinement. The numerical error δ_{RE} per Eq. (2) can be written as

$$\delta_{RE} = \frac{\varepsilon_{21}}{r^p - 1} \tag{4}$$

The uncertainty is, then, estimated by multiplying the numerical error δ_{RE} with a factor of safety FS such as

$$U_{FS} = FS \cdot |\delta_{RE}| \tag{5}$$

Xing and Stern (2010) define a distance metric $P = p_{RE}/p_{th}$, where p_{RE} is the estimated (or the observed) order of accuracy calculated using Eq. (3) and p_{th} is the theoretical order of accuracy (e.g., 2 for a second-order solver), and derived FS as a function of P as

$$FS(P) = \begin{cases} 2.45 - 0.85P, & 0 < P \leq 1 \\ 16.4P - 14.8, & P > 1 \end{cases} \tag{6}$$

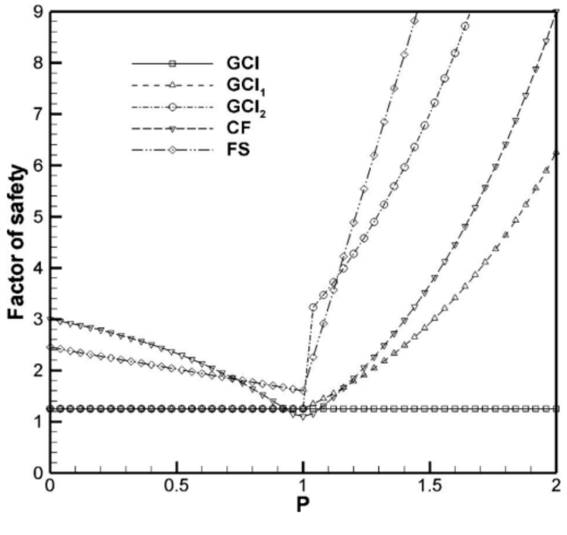


Figure 1. Factor of safety for different verification methods with p_{th} = 2 and r = 2 for the CF method (Xing and Stern, 2010).

Figure 1 shows the FS values over a range of P, with compared with other methods such as the Correction Factor (CF) method (Wilson et al., 2004) or the Grid Convergence Index (GCI, GCI1, GCI2) method (Roache, 1998). The authors state that the FS method is validated using statistical analysis of 25 samples with different sizes and argue that the FS method provides a reliability larger than 95%.

The Method by Eça and Hoekstra. This method determines ϕ_0, α, and p, by curve-fitting the solutions using the Least-Square-Root (LSR) method, again with the higher order terms ignored. Thus, this method requires at least four grids for the LSR process. Here, the symbol ϕ replaces the S in Eq. (1) to follow the authors' nomenclature.

For the uncertainty estimation, first a judgement is made for the quality of the data fit by using a data range parameter defined as

$$\Delta_\phi = \frac{(\phi_i)_{\max} - (\phi_i)_{\min}}{n_g - 1} \tag{7}$$

where n_g is the number of grids used for the curve-fitting. Then, the uncertainty is defined as

$$U_\phi(\phi_i) = \begin{cases} F_S \epsilon_\phi(\phi_i) + \sigma + |\phi_i - \phi_{\mathrm{fit}}|, & \sigma < \Delta_\phi \\ 3\frac{\sigma}{\Delta_\phi}\left(\epsilon_\phi(\phi_i) + \sigma + |\phi_i - \phi_{\mathrm{fit}}|\right), & \sigma \geq \Delta_\phi \end{cases} \tag{8}$$

Here, σ is the standard deviation of the fit. The safety factor F_S = 1.25 is used if the error estimation is considered reliable with $0.5 \leq p < 2.1$ and if $\sigma < \Delta_\phi$, otherwise F_S = 3 is used. The error estimator ϵ_ϕ is selected among different choices of $\delta_{RE} = \alpha h_i^p$, $\delta_1 = \alpha h_i$, $\delta_2 = \alpha h_i^2$, or $\delta_{12} = \alpha_1 h_i + \alpha_2 h_i^2$, depending on the magnitude of the p value such that:

$$\epsilon_\phi = \begin{cases} \delta_{RE}, & 0.5 \leq p \leq 2 \\ \delta_1 \text{ or } \delta_2, & p > 2 \\ \delta_1, \delta_2, \text{or } \delta_{12}, & p < 0.5 \text{ or impossible} \end{cases} \tag{9}$$

For each case of Eq. (7), the error model that gives the smallest σ is chosen.

Comparisons Between Two Methods. Both methods use the asymptotic expansion Eq. (1) based on the generalized Richardson

Extrapolation and estimate the numerical discretization error such that $\delta \approx \alpha h^p$ as shown in Eq. (2). Thus, the convergence of δ as $h \to 0$ depends on p, or from Eq. (3), on the sign and magnitude of the convergence ratio,

$$R = \varepsilon_{21}/\varepsilon_{32} \tag{10}$$

Figure 2 illustrates four possible types of convergence based on R ranges and the corresponding error models used by the FS method. Figure 3 illustrates three different ranges of the observed order of accuracy p value and the corresponding error models used by the LSR method.

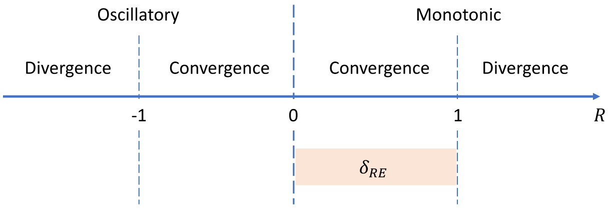


Figure 2. Convergence types based on the convergence ratio $\boldsymbol{R}$ and the corresponding error models used for the method of Stern et al. (the FS method).

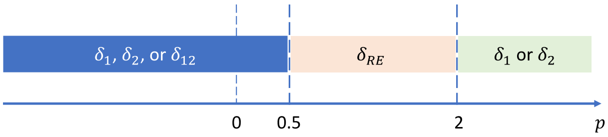


Figure 3. Different ranges of the observed order of accuracy $\boldsymbol{p}$ and the corresponding error models used for the method of Eça and Hoekstra (the LSR method).

The FS method is applicable only to the monotonic convergence type but is not applicable to all the other convergence types. This method uses the δ_{RE} in Eq. (4) as the error estimator. On the other hand, the LSR method directly uses the order of accuracy p value (instead of R) as a measure to select a proper error model from the choices listed in Eq. (9). Since the LSR method assumes that numerical simulations are of the 2nd-order or lower accuracy, thus limits the use of the error model δ_{RE} only to the cases of $0.5 \le p \le 2$. This p range corresponds to the case of $0.25 \le P \le 1$ in Eq. (6) of the FS method with $p_{th} = 2$. For $p > 2$, the LSR method again avoids using the δ_{RE} model since the order of accuracy is greater than the theoretical value of 2. Instead, the method introduces the δ_1 or δ_2 model to be used as an error estimator. Lastly, for $p < 0.5$, the LSR method introduces another error model δ_{12} that is a linear sum of δ_1 and δ_2, to allow a more flexible fitting of the possibly oscillatory data.

Table 1. Grid data and the simulation outcomes.

Grid, i	Total number of points	C_{FM} (e-3)	C_{TM} (e-3)
1	25,088,000	3.2148	4.0957
2	15,482,880	3.2116	4.0991
3	9,216,000	3.2075	4.1015
4	5,483,520	3.2021	4.1060
5	3,354,624	3.1945	4.1099

Table 2. Comparisons of VVUA results.

Var.	FS method (Grids 1, 3, 5)		LSR method (Grids 1~5)	
	C_{FM}	C_{TM}	C_{FM}	C_{TM}
p	1.69	1.12	1.72	0.72
δ_{RE}	-9.6e-3	1.3e-2	-9.3e-3	2.3e-2
$U(\%)$	0.5	0.6	0.4	0.7

An example is considered to compare the VVUA results by using the two different methods. Steady-state simulations are made for the model-scale frictional C_{FM} and total C_{TM} resistance coefficients of the Japanese Bulk Carrier (JBC) cargo ship by using a total of 5 different grids (Starke et al., 2024). Table 1 presents the grid data and the simulation outcomes, and Table 2 compares the VVUA results between the two methods. For the FS method, the grids 1, 3, and 5 are used, comprising a grid-triplet with a refinement ratio $r = \sqrt{2}$. The convergence ratio $R = \varepsilon_{31}/\varepsilon_{53}$ value is 0.57 for C_{FM} and 0.67 for C_{TM}, indicating monotonic convergence for both variables. For the LSR method, all five grids are used. The p value is 1.72 for C_{FM} and 0.72 for C_{TM}, thus $0.5 \le p \le 2$ for both variables and δ_{RE} is used as the error estimator ϵ_ϕ per Eq. (9). The resulting U values are comparable between the two methods with 0.4 ~ 0.5 % for C_{FM} and 0.6 ~ 0.7 % for C_{TM}.

Applications to practical problems such as high turbulent flows or complex geometries may be vulnerable to 'numerical noises' causing difficulties in achieving the so-called 'asymptotic range' required for a proper use of the RE approach. One of the criticisms of the FS method is that it can be sensitive to the spreads in the simulation results from grid sensitivity studies. An indication may be a higher value of the observed order of accuracy p far from the theoretical value, i.e., $P \gg 1$ in Figure 3. In this case, the large FS values may exceed such values that can be conceived as a typical 'safety factor'. As demonstrated from the above example, both methods, when the data exhibit monotonic convergence with the observed order of accuracy within expectations, provide practically equivalent error estimations. The FS method is simple to use and requires minimal computational resources among other methods. It can be a strategic approach first attempting to apply the FS method by using a grid-triplet. In case the convergence type of this data set is judged to be non-monotonic or non-converging, or the observed order of accuracy is out of the expectation, then one can proceed with using the LSR method. Nonetheless, it is noted there is criticism concerning the use of this method as it deals with even possibly diverging data without a good justification (Xing and Stern, 2015).

Future Updates. The present updates mainly focused on revisions for the verification process reflecting the user experience and suggestions for improvements learned through the survey. The major changes include a current set of definitions for VVUA, simplifying the mathematical derivations for a better readability, selecting the choices for different V&V approaches down from three to two, and providing step-by-step examples. Future updates should focus on revisions of the validation process with the most recent findings such as the N-version approach (Stern et al., 2017). Also, future updates should continue to adapt new verification processes providing better performance or demanding less computational resources.

Update of RP 7.5-03-01-02

Background. Based on the survey results, it was determined that there are no practical issues with this guideline, and a major update is not necessary. Therefore, we conducted a minor update, focusing on changes in terminology and the presentation of statistics.

Terminology correction. The term "total uncertainty" previously used is replaced with "statistics of comparison error". The reason for replacing the term is that in the previous uncertainty analysis, "total uncertainty" was defined as the combined value of the bias limits and the precision limits in EFD. However, in this guideline, "total uncertainty" was used with a different definition and meaning, which could lead to confusion.

The way to present the statistics of the comparison error. Due to insufficient detail in the guideline regarding how to present statistics of the comparison error, additional explanations have been included in this update.

4.4. Updates of other Procedures and Guides

The committee has also reviewed other guides per AC requests and made revisions/actions as summarized in Tab.3. (MC in table refers to the Manoeuvring Committee):

Table 3. Other guides per AC requests and made revisions/actions.

Number	Title	Actions
7.5-03-02-01	Uncertainty Analysis in CFD: Examples for Resistance and Flow	Minor changes
7.5-03-02-02	Benchmark Database for CFD Validation for Resistance and Propulsion	Database updated with new input
7.5-03-02-03	Practical Guidelines for Ship CFD Applications	No changes
7.5-03-02-04	Practical Guidelines for Ship Resistance CFD	Minor changes and corrections for inconsistencies in contents/chapters
7.5-03-03-01	Practical Guidelines for Ship Self Propulsion CFD	Minor changes
7.5-03-03-02	Practical Guidelines for RANS Calculation of Nominal Wakes	Minor changes

7.5-03-04-01	Guideline on Use of RANS Tools for Manoeuvring Prediction	Provided revision suggestions to MC
7.5-03-04-02	V&V of RANS Solutions in the Prediction of Manoeuvering Capabilities	Provided revision suggestions to MC

5. WAKE SCALING

The first SC on CFD and EFD Combined Methods identified a number of research topics within the area of combined EFD/CFD methods. These topics were ranked according to expected impact and possibility to improve with CFD. First in their ranking was the determination of the form factor, a subject they extensively studied and reported to the 2021 ITTC. Second in their ranking was the effective wake scaling and that subject has been studied by the present SC. CFD offers the possibility to compute wakes at either model or full-scale Reynolds number. Direct computation at full scale is attractive because it avoids extrapolation, but the primary drawback is the difficulty to estimate the quality of the solution since validation data is so sparse. Alternatively, it can be investigated if the assumptions underlying empirical methods, for instance the ITTC78 wake scaling formula, are confirmed by CFD computations. If so, it should increase confidence that CFD can be used as an alternative to the ITTC78 wake scaling formula, or that CFD can be used to improve the existing wake scaling formula.

If not, either the assumptions underlying the wake scaling formula are incorrect, or the CFD results are imperfect; that should then be further investigated. Here, the ITTC78 wake scaling formula and its components are shown as follows.

$$w_{TS} = (t + w_R) + \left(w_{TM} - (t + w_R)\right)\frac{(1+k)C_{FS} + \Delta C_F}{(1+k)C_{FM}} \quad (11)$$

Potential part: $(t + w_R)$

Viscous part: $\left(w_{TM} - (t + w_R)\right)\frac{C_{FS}}{C_{FM}}$

Roughness allowance part:

$$\left(w_{TM} - (t + w_R)\right)\frac{\Delta C_F}{(1+k)C_{FM}}$$

The first two parts consider the scale effects on the wake based on theoretical assumptions. The third part accounts for the influence of the surface roughness of an actual ship. All three parts are crucial for considering the full-scale wake, but to proceed cautiously in a step-by-step manner, we will limit our investigation here to the first two parts—the dominant factor in wake scaling—which are based on the respective works of Sasajima-Tanaka and Dickmann. Sasajima and Tanaka proposed that the nominal wake consists of a potential part and a viscous part; the potential part is the same for both the model and the ship, but the viscous part is subject to a scale effect. Dickmann proposed that the potential part of the wake fraction is proportional to the potential part of the thrust deduction. Detailed description of the full-scale wake formula can be found in RP 7.5-02-03-014.

- The three assumptions in the potential part and the viscous part of the ITTC78 wake scaling formula are:
- The potential part of the wake fraction is proportional to the potential part of the thrust deduction.
- The scale effect on the frictional part of the thrust deduction is small.
- The scale effect on the wake fraction is principally determined by a function of the scale effect on the friction coefficient.

To investigate if the validity of these assumptions is confirmed by CFD both model and full-scale CFD predictions have been made for 14 different ships, analysed by 9 different institutes and 7 different CFD codes. Potential wakes have been determined by replacing the viscous no-slip boundary condition at the hull by a free-slip boundary condition.

To maximize the number of cases that could be considered, the CFD set-up was simplified to double-body computations without the rudder, while surface-roughness effects for the full-scale ship were not taken into account. It was expected that such an approach would give a good first indication if the assumptions in the ITTC78 wake scaling formula are confirmed.

Figure 4 through Figure 6 give the CFD results that illustrate the three assumptions listed above. From these the following conclusions can be drawn:

- According to the CFD, the potential part of the wake fraction is indeed proportional to the potential part of the thrust deduction, with a (Pearson) correlation coefficient equal to r = 0.939.
- According to the CFD, the scale effect on the frictional part of the thrust deduction is not necessarily small. However, the frictional part is small compared to the potential part of the thrust deduction, it may not be an important assumption and thus may introduce a relatively small error in the extrapolation procedure.
- CFD does not at all support the assumption that the scale effect on the wake fraction is principally determined by a function of the scale effect on the friction coefficient: there is a large variation in predicted model-to-ship wake-fraction ratios between the various cases.

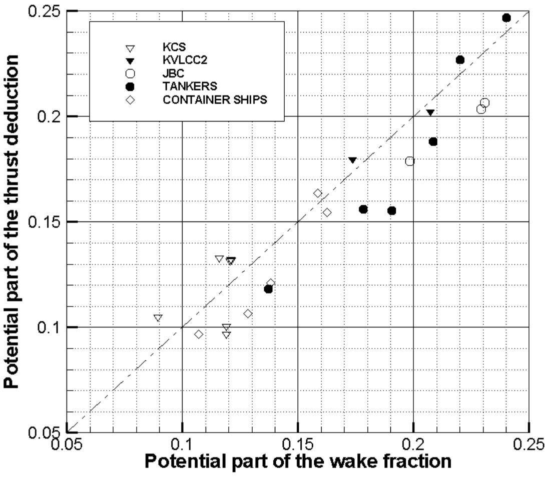


Figure 4. CFD predicted relation between the potential part of the nominal wake fraction and the potential part of the thrust deduction.

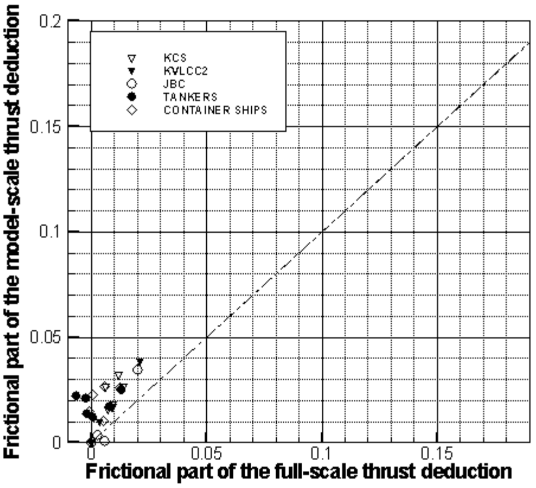


Figure 5. CFD predicted scale effect on the frictional part of the thrust deduction.

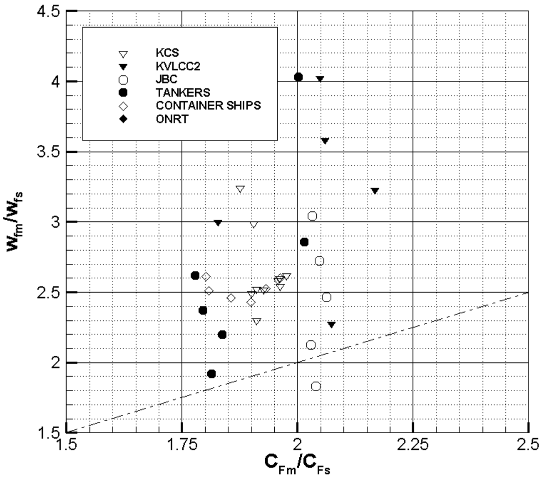


Figure 6. CFD predicted relation between the potential part of the nominal wake fraction and the potential part of the thrust deduction.

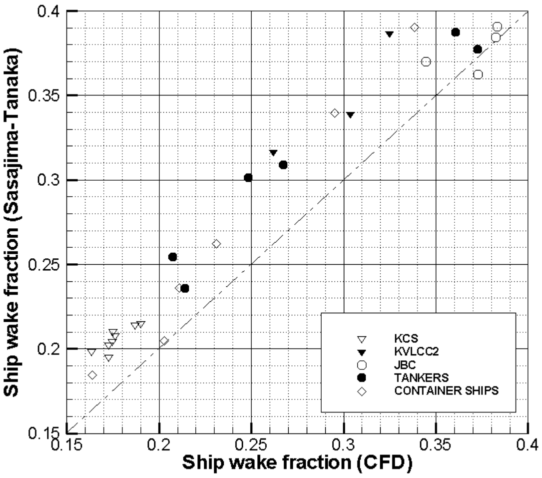


Figure 7. CFD-predicted nominal ship-wake fractions compared to the Sasajima-Tanaka method.

To investigate how the deviation from the three assumptions affects the prediction of the full-scale wake fraction, the CFD results have been used as input for the Sasajima-Tanaka method which targets nominal wake cases and the resulting full-scale nominal wake fractions have been compared to the nominal wake fractions coming directly from the full-scale CFD computations. In Figure 7 it can be seen that the full-scale CFD consistently predicts lower ship-wake fractions compared to the Sasajima-Tanaka method, with an almost constant vertical offset, except at higher wake fractions. The Pearson correlation coefficient evaluated for the data in this figure is r=0.976.

Contradictory to the Sasajima-Tanaka method, the ITTC78 wake scaling formula is based on the *effective* wake fraction rather than the *nominal* wake fraction. Additionally, the ITTC78 wake scaling formula includes the roughness allowance part. CFD propulsion computations have been performed for the same cases and the CFD predicted effective wake fractions have been compared to wake fractions determined with the ITTC78 wake scaling formula, using the corresponding model-scale CFD results as input. From Figure 8 it can be seen that this again results in lower wake fractions from the CFD compared to the extrapolation procedure with an almost constant vertical offset across the entire range of wake fractions without exception.

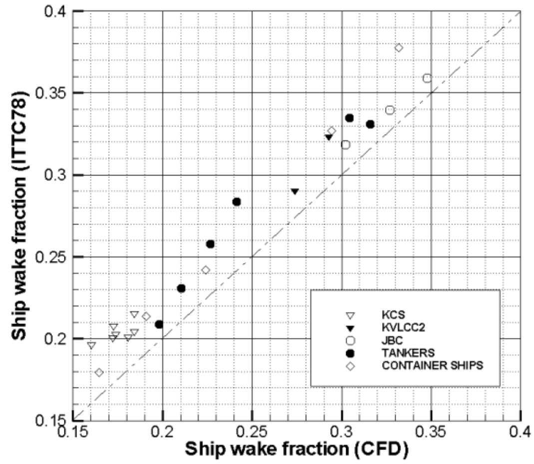


Figure 8. CFD-predicted effective ship-wake fractions compared to the ITTC78 wake scaling formula.

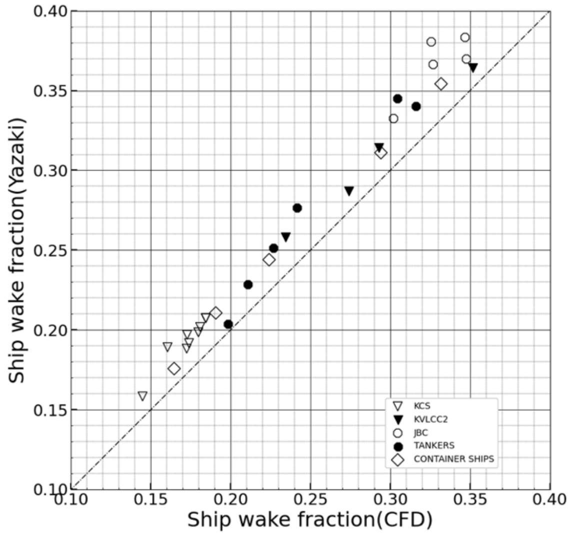


Figure 9. CFD-predicted effective ship-wake fractions compared to the Yazaki method.

And similar results are found when using the Yazaki method (1969), Fig. 9, which is based on correlation with trial data including roughness effect. It is concluded that there is a good correlation between the CFD predicted wake scaling and various empirical methods. Simplifications in the present CFD results will have some effect on the final wake scaling, and thus require further attention. Nevertheless, from this study the committee concludes that improvements in extrapolation procedures can be obtained, especially by replacing the assumed linear relation between the scale effect on the friction coefficient and the scale effect on the wake fraction with a CFD-based procedure.

Various combined EFD/CFD wake scaling procedures can be defined. For instance, replacing the friction ratio that is often taken from the ITTC57 line with a CFD predicted ratio between the wake fractions, or taking the ratio from the CFD predicted wake fractions to multiply the model-scale wake fraction that follows from, for instance, a wake survey:

$$w_{TS} = (t + w_{\mathrm{R}}) + \left(w_{T\mathrm{M}} - (t + w_{\mathrm{R}})\right)\frac{w_{\mathrm{S}}^{CFD}}{w_{\mathrm{M}}^{CFD}} \quad (12)$$

$$w_{TS} = w_{T\mathrm{M}}\frac{w_{\mathrm{S}}^{CFD}}{w_{\mathrm{M}}^{CFD}}$$

(13)

The committee recommends that a combined method for wake scaling be further analyzed, and any new method be evaluated using sea trial data.

6. OVERVIEW OF THE BENCHMARK CASES AND ONGOING CAMPAIGNS

This section deals with a comprehensive review of the benchmark campaigns relevant to CFD community which have been pursued in the last years in the naval hydrodynamic framework of interest for the ITTC reference community, with a particular focus on ship resistance and propulsion. This activity comes from a request of the previous committee, which considers the monitoring of the benchmark cases as a mandatory aspect to foster the growth of the trustworthiness of CFD calculations. Two Term of References (TORs) were assigned to the committee in the last ITTC conference (as reported in the introduction). The first one (TOR 4) focuses on the review of the outcomes of ongoing CFD benchmark campaigns, and support committees interested in organizing future benchmark activities. The second one (TOR 8) is focused on the definition of a standard procedure when a new benchmark case is planned.

6.1. Progress in the benchmark campaigns

To better understand the progress in the CFD capability in the marine research field, it is fundamental to monitor the evolution of the benchmark campaigns worldwide; therefore, within this section, the focus is on the analyses of the main outcomes arising from the latest CFD benchmark campaigns. Even if most of the sanitary problems concerning the SARS-CoV-2 (COVID-19) were generally overcome, this event reduced international cooperation in the last few years. Consequently, some of the planned international benchmark studies have been reduced in numbers and often postponed. One of the most emblematic examples is the Wageningen Workshop (which continues the Gothenburg and Tokyo Workshop series (Larsson, 2014 and Hino, 2020)), which was announced to be held in 2020 but has been postponed until 2025, so it is out of the scope of the present report.

Nevertheless, in the last four years, some benchmark campaigns have been monitored and found.

JoRes Project. The Joint Research (JoRes, 2024) project for "Development of an industry recognised benchmark for Ship Energy Efficiency Solutions" was a joint research project with more than 55 partners representing worldwide class societies, shipyards, research institutes, universities, propeller and Energy Saving Devices (ESDs) designers, CFD developers and CFD practitioners. The main objective of the project was the encouragement to establish ship full scale benchmark cases provided to the industry serving as a basis to validate the numerical results from CFD studies. Within the project time (2021 to 2024) two different vessels (a multi-purpose vessel and a tanker) have been investigated in several CFD workshops and results have been compared to the corresponding sea trial measurements. In addition to this, valuable CFD work of a flat plat benchmark case has been performed among the participant as well investigating details of different roughness models in full scale. Further full-scale data of four more vessels (ro-ro-ferry, cruise liner, tug and bulk carrier), suitable to serve as a benchmark case, have been collected within this project, but no CFD workshop has been made on this yet.

Considering the industrial nature of the project, most of the outcomes are still covered by a non-disclosure agreement (NDA); the consortium announced that the results will be shared and made available by the end of 2024.

The six different hull geometries, covering a wide range of worldwide ship types (multi-purpose-vessel, tanker, ro-ro ferry, cruise liner, tug and bulk carrier), some equipped with specific ESDs, have been collected. Valuable measurements at full scale (sea trials) were available for these geometries, whereas data at model scale were available only for some of them. In addition, to reduce the input data uncertainties, also some roughness measurements for the hull and propeller were provided and, for one case, full-scale flow measurements (PIV) were available of the inflow wake to the propeller at working conditions for a tanker. This quite rare data could be a valuable insight into a deeper understanding of effective wake at full scale and scale effects. The quality and variety of the data makes this activity very interesting; moreover, due to the well-known (strong) Reynolds number dependency on the ESDs performances, the availability of measurements at both model and full scale is deemed to deeply assess the CFD capability to predict the real efficiency of these devices. Considering the aim was to develop a full-scale benchmark case, also an uncertainty analysis on the sea trial measurements was reported, assessing an overall uncertainty level of about 4-6%. A first draft of the results was published on Ponkratov and Wheeler (2024). They report the evidence provided by one of the participants for a single test case, highlighting that the discrepancies for the predicted sea trials, with respect to the measurements, were about 4-6% for the shaft torque and power, and lower than 1% for the propeller revolution rate. This, as suggested by the authors, demonstrates that the actual CFD capabilities (even if with a not negligible computation effort: 40 million cells for about 100hours on 200 cores) are comparable with the sea trials.

The JoRes community established during this project will not end its collaborative work on analyzing full scale benchmark cases with CFD. Starting in 2024, CFD workshops will be organized by members of this consortium using the full-scale result data of the four vessels mentioned before (ro-ro-ferry, cruise liner, tug and bulk carrier).

Open Workshop on Full Scale Resistance Prediction: The Chalmers University of Technology has organized a freely available CFD benchmark campaign (workshop) in the second quarter of 2024, which has been shared among international research groups. The results will be available by the end of 2024. The workshop is focused on the assessment of CFD capabilities (blind test) in the estimation of full-scale resistance of an un-propelled ship advancing in calm water. The assessment will be pursued by comparing CFD submissions from different participants with full-scale experiments at varying speeds. The novelty of this activity with respect to the past, is the availability of full-scale experiments carried out without a propeller, therefore in towed conditions, so very close to resistance tests than self-propulsion trials. This will allow to focus the comparison on the resistance estimation without introducing any additional source of uncertainty coming from the presence of a propulsion system. The benchmark vessel is a 58m ship length (designed for a single screw-propeller) with a displacement of about 400 tons. The workshop is divided into three main stages, with mandatory and optional submissions. The first set of computations was performed at a fixed attitude in double-model (i.e. without free surface effects) using a mesh provided by the organizers. These simulations were used to compare the accuracy of different codes, setups, and schemes on a common grid. In the second set of computations, free surface effects were introduced; estimations were requested at different ship speeds. With the aim of focusing the comparison on the resistance (and its components) prediction, the computations were pursued at fixed (provided) sinkage and trim. Each participant has been requested to follow his own procedure and best practices, providing a comparison of the expected accuracy of these kinds of CFD simulations. Finally, the third sets were aimed at the analysis of the capabilities of the CFD predictions in estimating Reynolds number effects; to this aim, computations were requested for different scale factors for the reference speed. Even if this resistance benchmark campaign can provide very interesting data to assess the current CFD capability in the full-scale ship resistance prediction, the main outcomes cannot be reported here because their results will be shared only at the end of the 2024 year.

Benchmark on Scale Effect for Open-water Propellers. The liaison activities of this committee with the other technical groups within ITTC committees allows the creation of a new benchmark campaign. The ITTC Resistance and Propulsion Committee organized a benchmark activity devoted to assessing the impact of the model-to-full scale extrapolation on open-water propeller performances. The focus concerns the collection of CFD data useful to verify the accuracy of the actual ITTC extrapolation procedure and to collect relevant data suitable for possible improvement or to better assess its limits. The organizers adopt the well-known benchmark propellers provided by Schiffbau-

Versuchsanstalt Potsdam (SVA): the VP1304, named Potsdam Propeller Test Case (PPTC) for the SMP'11 Abdel-Maksoud (2011) and SMP'15 Workshops Kinnas (2015) and P1727, a controllable pitch and an unconventional tip rake propeller, respectively. The geometry of these propellers, already used in previous benchmark tests, was shared with the testing conditions at both model and full-scale. A matrix of CFD calculations at model and full scale was required at 5 advance coefficients fixing the propeller revolution rate (a single value at full scale and four values at model scale). All the participants were encouraged to adopt their best practices to perform the simulations and to collect their data following the proposed standard procedures. The results were collected by the end of March 2024, and the summary was shared during the ITTC 2024 conference after the drafting of the present report.

SIMMAN 2020. SIMMAN 2020 was the 3rd Workshop on Verification and Validation of Maneuvering Simulation Methods. The previous ones were organized in 2008 and 2014. Originally, the plan was to organize the workshop in 2019, but the project suffered from a four-year delay, due to the pandemic issues for COVID-19 (it was held in July 2022). The purpose of the workshop was to benchmark the capability of the ship manoeuvring simulators through comparisons with EFD results for different hull forms; namely, the three well-known ships widely used for several EFD and CFD benchmarks: the KRISO Container Ship (KCS), the KVLCC2 and the ONRT. The comparisons involved both free-running model tests and captive model tests. This benchmark study is noteworthy by the present committee because CFD-based methods were widely used to feed the systems-based model (virtual captive model tests) or to directly tackle the fluid dynamic problem (virtual free running tests). The workshop was organized by the Korea Research Institute of Ships & Ocean Engineering (KRISO) and The Society of Naval Architects of Korea and held in Incheon, Korea, in early June 2023. The workshop covers not only deep-water conditions but also shallow water ones up to a water depth to draught ratio of 1.2. For all the models, proper captive model tests were considered in shallow and deep water (except for ONRT, where only deep water was available), and, in the same conditions, experimental free model tests were performed. For the KCS and the ONRT, some trajectories were also recorded in wave (deep water) conditions. This latest data was the main novelty for this workshop with respect to the previous editions. Some main conclusions for the captive model tests are drawn from the workshop, see Quadvlieg et al. (2023a) and Quadvlieg et al. (2023b). EFD data are affected by a large scatter and some components cannot be compared among the measurements. The authors suggest improving the EFD measurements among all the institutions providing also an uncertainty quantification. The average error for the CFD-based predictions is generally lower than the empirical ones and the error goes from 6% for pure drift test growing for combined yaw and drift cases. The higher the value of the yaw rates, the larger the discrepancies will be (up to 20%). These conclusions are valid for both deep and shallow water conditions, but the shallow water presents slightly higher differences (10%). For the free running model tests, the main conclusions are that the scatter by CFD and empirical models is still comparable, and the discrepancies are significant, without being able to define which is the best practice for assessing a CFD prediction. The authors suggest further investigation in a new campaign.

NATO-AVT-STO. It is worth mentioning the activities that are pursued within the NATO Applied Vehicle Technology-Science and Technology Organization (AVT-STO). Several benchmark activities have been done and are currently ongoing concerning both physical studies and CFD/EFD assessments. Regarding maritime field, most of the activities have been focused on the assessment of the capabilities of CFD tools in predicting naval hydrodynamic related problems. However, the accesses to the results and the data collected are restricted to the NATO member states (with AUS and JAP in addition as extended opportunity partners), therefore, only few examples of activities are reported here. Activities focused on CFD and modelling assessment range from the analysis of turbulence and transition to turbulence model for CFD (Unsteady Reynolds-Averaged Navier-Stokes (RANS) based) simulations, such as the AVT-313 *"Incompressible Laminar-to-Turbulent Flow Transition Study"*. In these research activities, the assessment of the transition model for RANS code has been carried out on simplified geometries (such as flat plates and profiles) and at relatively low Reynolds numbers. In the AVT-301 team "*Flowfield Prediction for Manoeuvring Underwater Vehicles*", the capabilities of RANS based tools in the prediction of the flow field around complex geometries (the focus was a fully appended un-propelled submarine) have been assessed. The considered test cases concerned static manoeuvres of a submarine (straight ahead, static drift and steady turn), for which the flow field was characterized by largely separated flows, and a strong vortices/boundary layer interaction. Several submissions have been reported (with different solvers, different turbulent model, different computational meshes), allowing a comprehensive comparison between numerical predictions, including the influence of grid resolution and turbulence model on the estimation of local quantities and integral quantities. Ad-hoc experimental tests have been also conducted, providing a valuable data set for CFD benchmark. The conducted analysis highlighted a large discrepancy in the prediction of the flow field around the control surfaces (eventually stalled); this led to the need to deepen the analysis of CFD capabilities in the prediction of the flow around control surfaces at high Reynolds number (order of several Million) at large deflection angle (then eventually at stall conditions). This is the topic of the newer ongoing EFD/CFD benchmark study "*Assessment of numerical methods for complex flow over marine control surfaces*". Other benchmark studies in the marine hydrodynamic field have been focused on the flow field around surface vessels undergoing manoeuvres (both in calm water and in waves). For a complete view of these activities the interested reader can be referred to the public AVT-STO home page (https://www.sto.nato.int/).

Medium-scale Ship case. The Committee noted that China is planning to carry out flow field measurements on a medium-scale existing test ship. The ship was newly built in 2023 and it is mainly used for the validation of integrated energy-efficient propulsion systems, navigation technologies, and other new technologies. The main ship dimensions are: 55 meters in length, 10 meters in breadth, and 2.6 meters as design draught at 1500 tons in displacement (see figure 10). Currently, sea trials of self-propulsion and manoeuvring measurements have been carried out, but the data will be available only in the last part of 2024. In the first phase, the sea trials were conducted at four different propulsion conditions based on the ESD considered (see figure 11, i.e., without ESDs, with a Hub Vortex Absorbed Fins (HVAF), with a Pre-Swirl Duct (PSD), with a HSVF and a PSD. The self-propulsion tests were repeated three times without ESDs showing that the repeatability of the tests was about 1%. The manoeuvring tests, including turning circles and zig-zag, were conducted without ESDs, with a HVAF, and with a PSD. The novelty of this activity is the possibility of having consistent data without and with different energy-saving devices. In addition to the typical self-propulsion data, interesting 2D-3C PIV flow measurements of the inflow velocities to the propeller will be assessed. Unfortunately, due to the impact of COVID-19, the PIV measurement trial has been postponed to the second half of 2024, meaning these data are expected to be available in this committee's next term.

Figure 10. Overview of the medium-scale ship

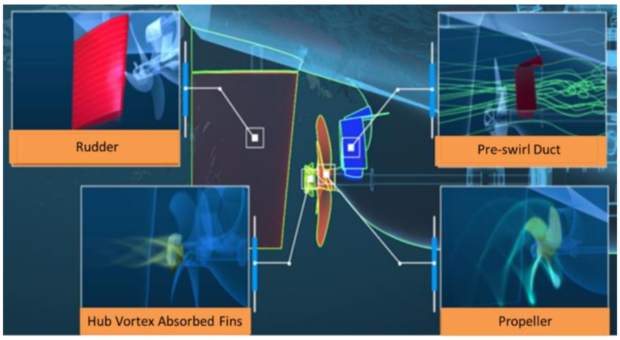


Figure 11. Propulsion system of the medium-scale ship

Future benchmark cases. The committee is also aware of some new benchmarks that will become available in the next years, but no detailed descriptions of them are currently available. Hereafter a not exhaustive list:

5. VTT (Finland) is working in the definition of a medium-size arctic ducted propeller-thruster case. The 'VTT Arctic Thruster' case involves model scale results in open water, full scale CFD including cavitation (OpenFOAM) and model scale CFD (OpenFOAM). The work has been conducted in the national MODPROP project funded by Business Finland. The thruster case will be published including the Computer Aided Design (CAD) model, open water experimental results together with the CFD tutorials.

6. SWOPP, organized by Rise (formerly SSPA) the project will provide a benchmark on wind assisted ships;

7. Wageningen 2025 CFD workshop (successor of Tokyo 2015 CFD workshop) is planned in summary 2025.

The committee recommends continuing to monitor the international scenario to be aware of additional data to be shared with the community.

6.2. Development of new benchmark cases

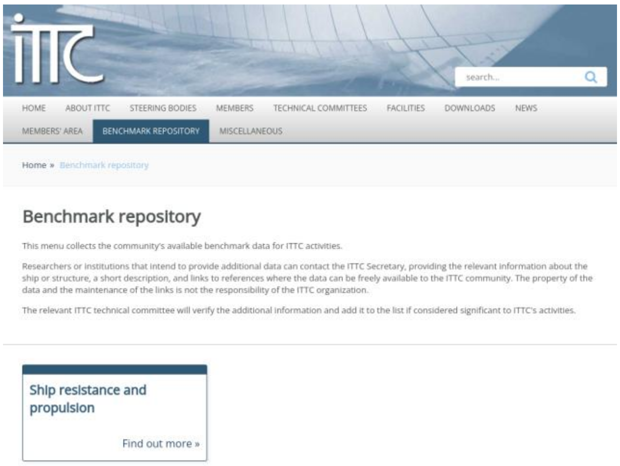
The second goal for this activity was the definition of a guideline (or a standard procedure) to generate a new benchmark case. Considering that the main interests of the marine engineer are in the ship resistance and performance predictions in general, they cover many different fluid-dynamic aspects, such as ship drag, self-propulsion, manoeuvring, seakeeping, and so on. All these classical marine problems are nowadays affordable by means of CFD tools as long as an adequate HPC infrastructure is available. Nevertheless, the key point to be able to trustfully tackle one of these kinds of simulations is the knowledge of reliable EFD similar data to compare with. This role is generally covered by the benchmark test cases. All the benchmarks share some general properties: they should be complete (all the data should be available) and well described in all their parts (all the data should be avoided by misleading). Apart from these general assumptions, each benchmark endeavour and experiment are unique, requiring proper ad-hoc procedures to be developed. For this reason, the present committee decided to avoid trying to standardize this process because the consequence could be an unused guideline or a too basic one. The only recommendations in this direction for a researcher who wants to assess a new benchmark can be sensitized in these simple questions:

- Did you completely describe how you performed the experiment?
- Did you provide all the data to reproduce it by other researchers?
- Did you report how you collect and analyses the results?
- Did you provide the level of accuracy of your input and output measurement?
- Are your data always available for the community?

These simple sentences can generally be a good way to guarantee that a benchmark test case can be used by the research community, particularly for CFD validation.

Nevertheless, in literature nowadays, considering that research institutes often test the same model more times (same geometry by different scholars or repeated tests with new equipment), similar (but not the same) data can be found from different sources. This often causes confusion and makes the collection of reference data a bottleneck in the CFD validation. For this reason, and in line with the previous conference proceedings, where the website was updated with the new “benchmark repository” section, the committee decided to collect all the sources of literature data (at the moment as references and links) in the ITTC survey repository, with the aim to share in a simpler and more effective way, all the nowadays available benchmark data (with a focus on the ship resistance and propulsion). This repository, which starts from the RP 7.5-03-02-02 material, collects 15 hull forms ranging from the historical ones (such as Wigley hull) to the most recent ones. The list collects systematically the ship name (or the used abbreviation), a short description of the vessel with the main characteristics, a list of links where the data can be downloaded and the references where the complete data was published. The Quality System Group, during last conferences, supported the idea to provide a structured space within the ITTC website fed by the community and under the supervision of the ITTC committees, to collect all the benchmark cases in all the marine field. The present committee agrees with this idea, but the activity was stopped by propriety issues. However, this repository, even if not still directly collects the data, can be considered a starting point for sharing data in this field, encouraging new researchers to compare and share their own data. This repository, if supported over the years, can encourage the naval hydrodynamic community to enlarge the database to be used to assess CFD capabilities and make the CFD predictions to be a more and more reliable tool for research and industrial activities.

Thanks to the contribution of the ITTC secretary, the committee uploaded and updated a webpage in the official ITTC site (https://ittc.info/benchmark-repository/ship-resistance-and-propulsion/). A new tab, named Ship resistance and propulsion, was added to the Benchmark Repository (previously under construction), as shown in Figure 12. In this page a link to a list of cases is reported (see Figure 13).



Benchmark repository

This menu collects the community's available benchmark data for ITTC activities.

Researchers or institutions that intend to provide additional data can contact the ITTC Secretary, providing the relevant information about the ship or structure, a short description, and links to references where the data can be freely available to the ITTC community. The property of the data and the maintenance of the links is not the responsibility of the ITTC organization.

The relevant ITTC technical committee will verify the additional information and add it to the list if considered significant to ITTC's activities.

Figure 12. Example of the new ITTC webpages.

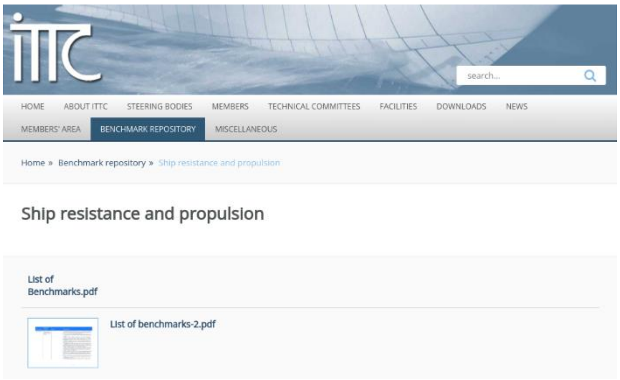


Figure 13. Example of the new ITTC report.

To further fosters the community to increase the available data with also more modern hull forms and propeller shapes, few lines are reported in the main pages, as follows: Researchers or institutions that intend to provide additional data can contact the ITTC Secretary, providing the relevant information about the ship or structure, a short description, and links to references where the data can be freely available to the ITTC community. The property of the data and the maintenance of the links is not under the responsibility of the ITTC organization. The relevant ITTC technical committee will verify the additional information and add it to the list if considered significant to ITTC's activities.

In light of this spirit, this committee suggests including in all the committees' TOR a request to check the correctness and the availability of the benchmark links and references in the list, updating when necessary. In addition, new committee members (each one with its own expertise) shall be involved in the verification part of the new community proposed cases.

7. LITERATURE REVIEW AND STATE OF THE ART

7.1. Advances, Accuracy and Challenges on Speed/Power Predictions

This section monitors and reviews the advances, accuracy and challenges within full-scale and model-scale CFD of maritime applications with special focus on speed/power pre-dictions.

The model-scale CFD methodology has been verified and validated for decades and reached an acceptable maturity level in terms of resistance, self-propulsion, and local flow predictions (Hino et al., 2020). However, the verification and validation effort for the full-scale CFD computations is relatively new and lagging behind mainly due to a lack of publicly available full-scale data. Lloyd’s Register conducted a workshop on ship scale hydrodynamic computer simulation with a blind test case for full-scale numerical modelling validation in 2016. The results of the workshop offer valuable insights into the performance of various numerical modelling techniques. Upon closer analysis of the published results, it was found that the mean comparison error for the predicted power was 13% for all submissions, indicating a certain degree of variability in the accuracy of the models. Nevertheless, it is worth noting that three out of the twenty-seven participants achieved highly accurate results, with errors below 3% for all considered speeds. Andersson et al. (2022) conducted a CFD benchmark study comparing industrial state-of-the-art ship-scale CFD predictions of the power reduction through installation of a Pre-Swirl Duct (PSD), where the objective was to both obtain an indication on the reliability in this kind of prediction and to gain insight into how the computational procedure affects the results. The 10 participants conducted 22 different predictions of the power reduction through a PSD installation on KVLCC2. The predicted power reduction varies around zero, on average 0.4%, with a standard deviation of 1.6%-units, if not considering two predictions based on model-scale CFD and two outliers associated with large uncertainties in the results. However, since there were no sea trial results existing, only comparisons of the differences in computational methods were conducted. In addition, recent publications, such as Sasamoto et al. (2020), Sun et al. (2020), Orych et al. (2021), Schouten et al. (2022), and Mikulec and Piehl (2023), demonstrated good accuracy of predicting sea trial conditions with full-scale CFD. However, most of the full-scale validation studies in literature are performed on a ship with one or a limited number of sea trials. As discussed in Korkmaz et al. (2021), a large number of sea trials are required for full-scale validations since the uncertainty of each trial is large.

Korkmaz et al. (2023) conducted an extensive comparative study between full-scale CFD-based ship performance predictions and sea trial results, following procedures compliant with the Safer and Cleaner Shipping (IACS) recommendation for evaluating EEXI values using CFD simulations within the established EEXI framework (IACS, 2022). The full-scale CFD simulations were carried out in accordance with their Best Practice Guidelines (BPG) developed based on the ITTC RP for the quality assurance in CFD (2021). Statistics of comparison errors of the full-scale CFD results were presented for delivered power and propeller turning rates based on 59 sea trial results from 14 vessels. While the accuracy of the CFD predictions may not seem too impressive, the authors argue that a significant portion of the standard deviation is due to scatter among the sea trials.

Liefvendahl et al. (2023) presented statistics of the delivered power ratio between 29 full-scale sea trials conducted on 15 vessels and those predicted by CFD in their RISE/SSPA report. The special motivation behind the creation of the report was stated to demonstrate that the procedures used at RISE/SSPA fully comply with the recent IACS guidelines for evaluating EEXI with CFD (IACS, 2022), so the contents of the report followed the guidelines of IACS.

Ponkratov and Wheeler (2024) pointed out that the main challenge is associated with the fact that sea trials procedures (like ISO15016) were not developed for the purposes of CFD validation. The main objective of these procedures is to confirm contractual speed. As a result, these procedures do not require hull and propeller roughness measurements (which are important for CFD) and still rely on simplified methods (sea state assessment by the naked eye, visual observations of vessel draughts etc). Clearly to develop an accurate case for CFD validation stricter requirements for the ship scale measurements should be implemented.

These expectations led to organizing and executing a JoRes joint research project aiming to develop an industry-recognized benchmark for ship-scale CFD validation. As discussed in Ponkratov (2023), comprehensive ship scale measurements for six vessels were performed including actual hull and propeller roughness checks. The propeller roughness was measured and estimated to be 4 μm. All the detailed measurement values and postprocessing details will be publicly available within the JoRes project benchmark until December 2024. These are preliminary sea trial measurements of JoRes consortium as shown in Table 4, compared with the CFD simulations. Overall, the simulations matched the sea trial favorably. Despite all the effort to perform sea trials as accurately as possible, as discussed in Ponkratov and Strujik (2023) the sea trials uncertainty level for this case was 4-6%. The results can be considered to be validated as the CFD results are within this range.

Table 4. Percent comparison of CFD results to full-scale sea trial measurements

Percent Comparison of CFD Results to Sea Trial Measurements		
Metric	11.25 knots	13.34 knots
Propeller Rotation Rate	0.29%	0.66%
Torque	-5.75%	-4.64%
Shaft Power	-5.43%	-3.69%

7.2. Turbulence Models and Wall Treatments at Full Scale

This section reviews and studies the performance of turbulence models and wall treatments at full scale. The development of new turbulence modelling approaches and their performance for marine applications are also introduced. The section focuses on two main areas: turbulence models at full scale and wall treatments at full scale.

RANS Turbulence models at full scale. It was challenging to simultaneously satisfy both Froude number (Fr) and Reynolds number (Re) similarity, necessitating a choice between gravity similarity and viscosity similarity based on the specific research focus. However, full-scale simulation, unlike traditional scaled-down approaches, allows for the simultaneous fulfilment of both gravity and viscosity similarity requirements. It is crucial to select suitable turbulence models and boundary conditions in full-scale simulations.

The RANS method stands out for its high computational efficiency, acceptable precision, and well-established development, making it currently suitable for full-scale simulations of ships and marine structures. More than 90% of the full-scale simulations are done by RANS method. Many studies demonstrate the capability of full-scale RANS numerical simulations on determining ship performance, such as powering prediction (Song et al., 2020), seakeeping performance evaluation (Niklas & Pruszko, 2019), wake scaling (Can et al., 2020), and hull optimization (Seok et al., 2019). Precision of several percent is documented by comparison with full-scale trial for powering prediction (Korkmaz et al., 2023).

With respect to the validation of full-scale RANS simulations, Eça et al. (2023) assessed the effect of the Reynolds number on the performance of six different RANS turbulence models in the simulation of viscous flows at $Re = 2 \times 10^6$ (model scale,) and $Re = 10^9$ (full scale). It was found that discrepancies between solutions of the six selected turbulence models at model scale Reynolds number were significantly larger than those observed at full scale Re. This indicates that modelling error assessments (validation) performed at model scale Reynolds number cannot be extrapolated to full scale. Moreover, the small discrepancies obtained between the solutions of six turbulence models at full scale is encouraging. The ability to simulate full scale conditions, for which there is almost no experimental data available, is one of the biggest assets of CFD.

Detached Eddy Simulation (DES) Turbulence models at full scale. A small subset of researchers, driven by the demands of their studies, has adopted for more precise 'DES-type' turbulence models in full-scale simulations, with particular attention given to the Improved Delayed Detached Eddy Simulation (IDDES) method (Gritskevich et al., 2012). To accurately simulate the full-scale ship boundary layer, three turbulence modelling strategies suitable for simulating ship flows — k-ε, SST k-ω, and IDDES — were evaluated and compared with full-scale ship propeller torque data from the 138-meter general cargo ship MV Regal by Pena et al. (2020). Among the three different models, the IDDES replicated the sea trial measurements with the highest accuracy. Additionally, the study confirmed that the choice of turbulence strategy significantly influences the full-scale velocity field at the stern of the ship, especially in the wake and jet regions. In addition, the influence of hull roughness on the wake was investigated at real scale with and without propulsion by Kanninen et al. (2022). As simulations progressed, the differences in the wave patterns at the ship's stern between simulations using RANS and DES turbulence models became more pronounced. DES demonstrated better performance in addressing turbulence and wave breaking issues.

Besides the RANS and DES models, Large Eddy Simulation (LES) model is another option for full-scale simulations. While, due to the rapid reduction in turbulent scales with increasing Reynolds numbers, LES is currently not suitable for engineering applications at high Reynolds numbers.

Other turbulence modelling methods. There are developments in the modelling of laminar-turbulent transition, most noteworthy for the assessment of (scale effects on) propellers in open water and in behind conditions. Numerous studies have demonstrated the necessity of incorporating transition modelling in CFD to accurately determine propeller performance at model-scale, whereas full-scale simulations can successfully rely on two-equation turbulence models.

Kerkvliet et al. (2024) demonstrated that CFD, when employing the appropriate turbulence and transition models, can accurately predict propeller performance at model-scale, in scenarios involving partially laminar or fully turbulent boundary layers. The findings provide additional insights on improving extrapolation methods through CFD simulations, particularly when model-scale considerations prioritize the accurate development of turbulent boundary layers. A modelling method on laminar-turbulent transition was proposed based on Reynolds averaged Navier-Stokes solver (RANS) in combination with the k-ω SST turbulence and γ-Reθ transition model for a modern designed MARIN stock propeller. The predicted results by the method were compared to Experimental Fluid Dynamics (EFD) results, which involved propellers equipped with and without innovative turbulence stimulators, also known as turbulators. In addition to comparing performance characteristics, the boundary layer flows regimes were also examined using EFD paint test results. Furthermore, full-scale Reynolds numbers CFD simulations were conducted and compared to conventionally extrapolated EFD results. Excellent comparisons were achieved between EFD and CFD for model-scale Reynolds numbers, encompassing both uncontrolled and passively controlled boundary layers. A clear trend of Reynolds scaling was observed for propellers with a turbulent boundary layer at model-scale. This study enhances the understanding of boundary layer behavior in model and full-scale propellers. These insights are essential for improving model-test accuracy and to enhance the accuracy of full-scale performance predictions, with as end goal to design more efficient marine propulsion systems.

Wall treatments at full scale. There are significant activities and publications on simulating the effects of surface roughness for the wall treatment at full-scale. Roughness effects are one of the main challenges of the prediction of full-scale ship resistance using traditional model tests and extrapolation procedures.

Ohashi (2021) conducted a numerical study of roughness effects at an actual ship scale. In this study, Low-Reynolds number roughness models are developed, based on the two-equation turbulence model. Meanwhile, a wall function method to account for the roughness effect is also developed based on the assumption of local equilibrium. Both models are examined with respect to the computation of the 2D flat plate case at the actual ship scale. The resistance coefficients of the low-Reynolds number models increase with the roughness height similar to the value of the empirical formula. The resistance coefficients of the low-Reynolds number models increase with the roughness height similar to the value of the empirical formula. The wall function method also works properly with changing roughness height and Reynolds number. The uncertainties in the resistance coefficient of the wall function method reach larger values than those of the low-Reynolds number models at the Reynolds number 1.0×10^7 and the uncertainties become smaller at higher Reynolds numbers. Additionally, the distributions of the non-dimensional velocities u+ based on the non-dimensional heights y+ of the low-Reynolds number models and the wall function method are compared for changing the roughness height.

Moreover, both the roughness models and wall function method are applied to simulate the wake flow before the propeller plane of a ship at full scale. The simulated velocity contours are compared to the measured result obtained from the sea test of the actual ship. Both simulated results show good agreement with the measured data. The uncertainties in the total and pressure resistance coefficients of the wall function method are slightly smaller than those of the low-Reynolds number models. Consequently, the wall function method is a better method for full-scale simulations with the roughness effects.

Eça et al. (2022) simulated the flows around different geometries (flat plate, submarine and two ships) at full scale Reynolds numbers (10^8 to 10^9) with RANS solvers using the k-ω SST eddy-viscosity model. Roughness effects are included in the k and ω boundary conditions for values of the sand-grain roughness height covering hydraulically smooth and fully rough surfaces. The results show that with the proper scaling, the increase of the friction resistance coefficient with the sand-grain roughness height is equivalent for the four geometries tested. Conversion of average roughness height to sand-grain roughness is assessed by comparing CFD results with empirical correlations. Simulation results show the best agreement with the Townsin et al (1984). correlation with a small variation of the ratio between average roughness and sand-grain roughness heights.

Upcoming workshops, such as JoRes JIP and the Wageningen Workshop, should provide relevant information about accuracy and uncertainty of full scale and model scale ship predictions.

7.3. Advances in the Application of Detailed Flow Measurements

This section reviews the recent developments and applications of Particle Image Velocimetry (PIV) as a tool to provide detailed flow measurements for ship flows. This will focus on three main areas of recent activity: full scale, hydrodynamic and aerodynamics measurements.

Full scale measurements. The increased demand for full scale validation data for ship resistance and propulsion has led to a number of benchmark campaigns in this area (described in more detail in section 5). In particular the JoRes project has conducted full scale PIV measurements which will be published later in 2024. The PIV method used in this benchmark campaign is detailed by Birvalski et al (2023) where stereo PIV measurements are conducted in the wake of a full-scale sailing ship. They measured part of the propeller inflow area including the peak of the viscous wake using a novel PIV device called the 'FlowPike'. A range of different ship speeds and sailing conditions were assessed in order to help improve full-scale ship CFD simulations in the future. (add figure with measured wake?). Ponkratov & Wheeler (2024) provide a summary of the progress made in full-scale validation data since 2015 and compare their CFD simulations with the JoRes1 tanker full scale data.

There are also planned full-scale PIV measurements as part of the CSSRC full-scale benchmark campaign to be conducted later in 2024. This data along with model scale PIV measurements that have already been conducted should be available at some point in the future.

Hydrodynamics measurements. Since 2020 there have been a range of PIV measurement campaigns focused on Ship hydrodynamics.

Sun et al. (2020) used PIV to measure the flow velocities between the main and demi-hulls of a trimaran and compare this to CFD simulations. One demi-hull was made of a polycarbonate material with 90% light transmission rate and a refractive index 1.58 (close to that of water 1.33) to avoid occlusions from the demi-hull blocking the light source/camera view. A non-standard calibration was performed, and the optical distortions were mapped and corrected.

Wu et al (2020) used PIV to measure the wake flow field behind a Panamax Bulker ship model in a ballast condition and compared it to the design condition. The time-averaged velocity, turbulent fluctuations, turbulent kinetic energy (TKE), Reynolds stresses, and vorticity information were measured indicating a significant difference at the top of the propeller plane in ballast condition due to the proximity of the free surface.

Ortolani et al (2020) investigated propeller off-design conditions associated with a twin-screw vessel in straight ahead and drift angles of ±13◦ and ±27◦. In each condition the propeller inflow condition was measured using a boroscopic-based Stereo-PIV system and compared to the measured forces on individual propeller blades.

Bhushan et al (2021) conducted tomographic PIV measurements in a towing tank for the first time to measure vortical structures in the flow around the surface combatant 5512 with static drift angles of 0, 10° and 20°. The TPIV measurements provided detailed measurements for the progression of the vortical structures and associated mean and turbulent flow characteristics for ship flows. However, they had larger uncertainties near the hull (up to 17%), which limits analysis of the vortex onset characteristics. The authors followed this up with new experiments using four-dimensional particle tracking velocimetry (4DPTV) (Sanada et al., 2023). The same model was used with a static drift angle and a pure sway condition. A comparison between the TPIV and the 4DPTV setup is provided along with the advantages and disadvantages of each.

Deng et al. (2021) used a stereo PIV system in a towing tank to assess the flow field around the bow of a trimaran hull form with different bow configurations, including different bulbous bow designs and T-foils. Model tow speeds between 1.766 and 2.943 m/s were used.

She et al. (2021) conducted time resolved PIV measurements on the water entry of a typical 2D bulbous bow section taken from the KCS hull. Drop tests were conducted to investigate the free surface interactions and measure the induced flow velocities and estimate the surface pressure distribution based of pressure reconstruction.

Guo et al (2021a) investigated the flow field around the bow of a scientific research vessel using flow visualization methods (such as tufts) and stereo PIV at multiple planes. A 3-dimensional (3D) three-component (3C) space reconstruction of the time-averaged flow field around the bow was created. This method was then used in Guo et al (2021b) to characterize the trajectory of bubbles in the flow and compared this with the flow direction from a simple tuft visualization.

Jacobi and Nila (2021) conducted synchronised PIV and Digital Image Correlation (DIC) experiments in a towing tank for the first time to assess the fluid structure interactions of a flexible hydrofoil. This allowed the fluid flow field to be characterized at the same time as full-field structural deformations.

Jacobi et al (2022) characterized the flow velocity around the bow of a fast ship using stereo PIV at Froude numbers up to 0.8. The obtained velocity field is subsequently used for a volumetric description of the time-averaged hydrodynamic pressure field, with the pressure reconstruction conducted in OpenFOAM. A similar methodology is used by Jacobi (2023) to reconstruct the three-dimensional flow and pressure field from a total of 28 PIV measurement planes for a hydrofoil at angles of attack ranging from 0-12◦ in steps of 3◦

Savio et al (2024) evaluated the hydrodynamic performance of a flexible marine propeller in a cavitation tunnel. The fluid structure interaction was characterized using both stereo PIV and Stereo DIC measurements to measure both the propeller deformations and the flow field downstream of the propeller.

Aerodynamic measurements

Alongside the hydrodynamics measurement campaigns, there have been several studies conducted into the aerodynamics around ships.

Correia et al (2021) used PIV to characterize the impact of ship stack geometries on exhaust smoke dispersion in ports using wind tunnel experiments.

Setiawan et al (2022) used PIV experiments to investigate the impact of a simulated Atmospheric Boundary Layer (ABL) on the aerodynamic flow field over the flight deck of two different Naval vessels. The results showed that the ABL increased the turbulent fluctuations but reduced the length of the wake along the flightdeck centreline.

Hysa et al (2023) presented a multi-directional redundant 3D-PIV system for ship deck wind interactions. They investigated the need for redundant imaging and illumination directions with current results showing both the measurement domain and data interpretability benefit from redundancy in both areas.

Zhu et al (2024) used a high-speed, dual-plane, stereoscopic PIV system to measure the aerodynamic flow field around a simplified frigate geometry, with the aim of estimating various other planes in the flow field using a reduced order model.

<u>Summary.</u> The range of published experimental data using or developing PIV methods for ship-related flows indicates that this is an active area of research. This includes the development of new methods that will increase the range of validation data available in the future. Key examples include full-scale ship wake data, the use of tomographic PIV and Particle Tracking Velocimetry to characterize vortical structures, the use of near transparent hulls for reducing visual occlusions and the synchronized use of PIV and DIC to provide fluid-structure interaction data for flexible propellers and hydrofoils.

8. LIAISON WITH OTHER COMMITTEES

The specialist committee has initiated several contacts and liaisons with other committees, groups and people inside and outside the ITTC.

<u>Contact to other chairman and chairwoman of ITTC committees:</u> One of the tasks of TOR1 was to get in contact with other committees to get informed about newly established and used numerical methods in combination with experimental procedures. Furthermore, the contact should encourage the other committees to look explicitly for new combined EFD / CFD methods in their field of topic.

Contact was made and replies have been received from Full Scale Ship Performance Committee especially on details of shallow water correction for sea trial evaluation, the need of reliable full scale performance data and the details of full-scale flow field data. The Specialist Committee on Cavitation and Noise has been contacted with no further relevant update concerning upcoming new methods but existing applications for calculating self-propulsion points in model and full scale as well as for the design of wake-generating dummy models for experimental cavitation analysis. A good contact has been established with the Specialist Committee on Ice where noticeable methods are arising (not yet widely used) simulating ice movement, ice interaction (hull, propeller, ice) and ice breaking. Challenges are here to validate numerical results.

Our committee has been contacted by the Seakeeping Committee to answer their specific question on V&V methods for sea keeping and related unsteady RANS flows.

<u>Review of procedures:</u> For the review of procedures, the committee has been in contact with the manoeuvring committee of the ITTC.

<u>Survey among ITTC members:</u> For the tasks in TOR 9, 10 and 11 the specialist committee set up a survey and distributed it to all members of the ITTC. The focus was on the usage of validation and verification procedures of the ITTC within the context of numerical calculations. The feedback was very satisfying and could serve as a valuable basis for the committee to update the procedures relevant for the V&V (TOR 13). The results of the survey and the corresponding update of RPs are described in more detail in section Survey on V&V and Quality Assurance in CFD.

<u>Review of IACS Recommendation No. 173:</u> The specialist committee has been contacted by the IACS working group and requested to review their new, at that time unpublished, procedure IACS No. 173 “Guidelines on Numerical Calculations for the purpose of deriving the V_{ref}

in the framework of the EEXI Regulation" (IACS, 2022).

Driven by the IMO's focus on reducing emissions from ships and improving their energy efficiency, the IMO has recently introduced the Energy Efficiency Existing Ship Index (EEXI) for all existing ships, regardless of when they were built (IMO 2022a, 2022b). In the newly established framework of EEXI regulations targeting existing vessels, it becomes acceptable to evaluate propulsion performance of ships using only CFD simulations without towing tank testing.

The guidelines, IACS Recommendation No.173 (2022), require applicants for determining EEXI values based on CFD simulations to demonstrate their qualification in quality assurance for CFD simulations according to ITTC RP 7.5-03-01-02. The IACS recommendation presents a numerical calculation methodology, which involves three steps: demonstration of qualification, validation/calibration and calculation. Through these steps, procedures are presented to determine the vessel's reference speed (V_{ref}) required for calculating the EEXI value solely through CFD calculations.

The committee discussed referencing the IACS guideline No. 173 from the ITTC RP 7.5-03-01-02 as an exemplary guideline to produce CFD results based on correlation factors derived on model tests or sea trials. The committee abstained from recommending this, as the IACS guideline is a detailed recipe for the usage of CFD in a special context and references itself on the more generally described basic guidelines of the ITTC in this sense.

<u>Contact to organizes of CFD benchmark campaigns:</u> For the investigation of the description of the different benchmark campaigns (see section, overview of the benchmark cases and ongoing campaigns) the committee has multiple contact with other organizers of such campaigns. For example, the organizers of the JORES project (Jores, 2024) have been contacted by the committee for a discussion. The organizers of the upcoming Wageningen 2025 CFD workshop have been contacted as well. With the Resistance and Propulsion committee of the ITTC contact has been established to receive information on a specific benchmark campaign.

9. FINAL REPORT AND RECOMMENDATIONS TO THE 30TH ITTC

The 30th ITTC Combined EFD/CFD Specialist Committee has made the following conclusions and recommendations:

9.1. General Technical Conclusions

<u>TOR1.</u> The committee performed a literature survey about combined methods. We contacted other committee chairs via email and in person (when possible) to inquire about new combined methods, and to encourage them to consider developing new methods.

The committee considered the future work plan from the previous ITTC, and selected what it thought to be the best candidate to develop a new method, which is to study wake scaling. The committee embarked on a study that explored the assumptions in common wake-scaling formulae (ITTC 1978, Yazaki *replace with references*), and made detailed comparison of wake predictions at full and model scale for a range of ship type, with a variety of CFD codes.

<u>TOR 2.</u> The committee was in contact with the other technical committees to follow and encourage new combined methods. A survey was distributed to request information about current, new, planned, and desired combined methods.

<u>TOR 3,4,5,6.</u> There is an increase of full-scale (and model-scale) CFD applications appearing in the literature. There are notable examples in the recent literature of using turbulence models and wall treatments at full scale. Many studies demonstrate the capability of full-scale CFD numerical simulations on determining ship performance, such as powering

prediction, seakeeping performance evaluation, wake scaling, and hull optimization. Precision of several percent is documented by comparison with full-scale trial for powering prediction.

There are significant activities and publications on simulating the effects of surface roughness at full-scale.

There are developments in the modelling of laminar-turbulent transition, most noteworthy for the assessment of (scale effects on) propellers in open water and in the behind condition.

Upcoming workshops, such as JoRES JIP and the Wageningen Workshop, should provide relevant information about accuracy and uncertainty of full scale and model scale ship predictions.

TOR 7. The range of published experimental data using or developing PIV methods for ship related flows indicates that this is an active area of research. This includes the development of new methods that will increase the range of validation data available in the future. Key examples include full-scale ship wake data, the use of tomographic PIV and Particle Tracking Velocimetry to characterise vortical structures, the use of near transparent hulls for reducing visual occlusions and the synchronised use of PIV and DIC to provide fluid-structure interaction data for flexible propellers and hydrofoils.

TOR 8. The committee decided to not develop a standard process for benchmark studies. The variety of studies is so great that a single process would have to be so general that it would not be useful for any particular study.

Instead, the committee worked with the ITTC Secretary to establish a single online location for benchmark studies to be represented on the ITTC website. The first version of collected benchmark cases for resistance and propulsion is online.

TOR 9. A literature study and a survey among members of the ITTC have been done on the use of V&V. More than 40 responses to the survey were received. Over 85% of the survey participants have carried out V&V studies. From the literature study, no relevant new methods for V&V, nor uncertainty assessment, have been found.

TOR 10. RP 7.5-03-01-01 has been substantially revised, including a new front section providing up-to-date definitions and objectives of CFD VVUA, down-selecting to two V&V methodologies, and step-by-step examples of the verification procedures. This update reflects the suggestions that were collected through the survey among ITTC members.

TOR 11. The committee conducted a survey to all ITTC members on the use of the Recommended Procedures 7.5-03-01-02 “Quality Assurance in CFD Ship Applications”. The results of the survey indicate that more than half of respondents are currently using the new RP 7.5-03-01-02, or something similar, for quality assurance.

A major revision has been deemed unnecessary at this time based on the survey results. Two minor updates have been made to the RP 7.5-03-01-02.

TOR 12. The members of the committee, together with several external collaborators, authored a paper in the 35th Symposium on Naval Hydrodynamics that presents results on the study of wake scaling. The paper analyses current wake extrapolation formulae and discusses how computations of ship wakes could be used in a future combined method. The paper also uses the new 7.5-03-01-01 procedure to assess uncertainty.

TOR 13. RP 7.5-03-01-01 and RP 7.5-03-01-02 have been updated according to feedback from the AC and the committee-distributed survey.

9.2. Recommendations

- The committee recommends that the list of future work proposals (including Appendix A, and those in Section xxx) be considered as a guide to continue the work on combined methods in the next ITTC.

- It is recommended that in the future a survey reviewing the use and desire for combined methods be distributed early in the committee work, and the feedback be incorporated in any plans to revise current and pioneer new combined methods.

- It is recommended that the benchmark repository on the ITTC website should be maintained and supported by the ITTC organization to guarantee the correctness and completeness of the information found therein.

- It is recommended to monitor the success of the ITTC benchmark repository and to periodically update it, if necessary. It is also recommended that each new committee consider expanding the list of benchmark cases in their respective fields.

- It is recommended to adopt the updated RP 7.5-03-01-01, and RP 7.5-03-01-02.

- The committee recommends collecting feedback on the usage of the interim version of RP 7.5-03-01-01, and to revise it based on the feedback. It is also recommended to include examples of validation in a subsequent update.

- It is recommended that RP 7.5-03-01-01 be extended for unsteady flows so that it can be used for seakeeping, manoeuvring in waves, and other unsteady problems.

10. REFERENCES

Abdel-Maksoud, M., 2011, “Proceedings of the workshop on cavitation and propeller performance”, The Second International Symposium on Marine Propulsors. SMP’11, Hamburg, Germany.

AIAA, 1995, “Guide for the verification and validation of computational fluid dynamics simulations”, American Institute of Aeronautics and Astronautics, AIAA-G-077-1998, Reston, VA.

Andersson, J., Shiri, A.A., Bensow, R.E., Yixing, J., Chengsheng, W., Gengyao, Q., Deng, G, Queutey, P., Xing-Kaedin, Y., Horn, P., Lücke, T., Kobayashi, H., Ohashi, K., Sakamoto N., Yang, F., Gao, Y., Windén, B., Meyerson, M.G., Maki, K.J., Turnock, S., Hudson, D., Banks, J., Terziev, M., Tezdogan, T., Vesting, F., Hino, T., Werner, S., 2022, “Ship-scale CFD benchmark study of a pre-swirl duct on KVLCC2 Numerical Ship Hydrodynamics”, Applied Ocean Research, Vol.123:103134.

ASME, 2009, “Standard for Verification and Validation in Computational Fluid Dynamics and Heat Transfer”, American Society of Mechanical Engineering, Standard No. ASME-V&V20.

Bhushan, S., Yoon, H. and Stern, F., 2021, “Detached eddy simulations and tomographic PIV measurements of flows over surface combatant 5415 at straight-ahead and static drift conditions”, Ocean engineering, Vol.238:109658.

Can, U., C. Delen, S. Bal, 2020, "Effective wake estimation of KCS hull at full-scale by GEOSIM method based on CFD", Ocean Engineering, Vol.218:108052.

Cavallo, P. A., and Sinha, N., 2007, “Error Quantification for Computational Aerodynamics Using an Error Transport Equation”, J. Aircr., Vol.44(6), pp. 1954-1963.

Cavallo P. A., Sinha, N., and O'Gara, M. R., 2008, "Viscous Error Transport Equation for Error Quantification of Turbulent Flows," 38th Fluid Dynamics Conference and Exhibition, Seattle, WA, Jun. 23-26, AIAA Paper No. 2008-3851, pp. 1-16.

Celik, I., and Hu, G. S., 2004, "Single Grid Error Estimation Using Error Transport Equation," ASME J. Fluids Eng., Vol.126(5), pp. 778-790.

Correia, L.P., Rafael, S., Sorte, S., Rodrigues, V., Borrego, C. and Monteiro, A., 2021, "High-Resolution Analysis of Wind Flow Behavior on Ship Stacks Configuration: A Portuguese Case Study", Atmosphere, Vol.12(3), p.303.

Deng, R., Wang, S., Luo, W. and Wu, T., 2021, "Experimental study on the influence of bulbous bow form on the velocity field around the bow of a trimaran using towed underwater 2D-3C SPIV", Journal of Marine Science and Engineering, Vol.9(8), p.905.

Eça, L. and Hoekstra, M., 2014, "A Procedure for the Estimation of the Numerical Uncertainty of CFD Calculations Based on Grid Refinement Studies", J. Comput. Phys., Vol.262(1), pp. 104-130.

Eça, L., Kerkvliet, M., Toxopeus, S.L., 2023, "Comparison of RANS Turbulence Models for the Simulation of Smooth Wall Boundary-Layers in Pressure Gradients at Moderate and High Reynolds Numbers", 10th Conference on Computational Methods in Marine Engineering, Madrid, Spain.

Eça, L., Starke, A.R., Kerkvliet, M., Raven, H.C., 2022, "On the contribution of roughness effects to the scaling of ship resistance", Journal of Ocean Engineering and Marine Energy, Vol.8, pp. 539-551.

Guo, C.Y., Kuai, Y.F., Wang, W., Han, Y., Xu, P. and Fan, Y.W., 2021a, "Experimental study on bow flow field of scientific research ship based on flow visualization", Ocean Engineering, Vol.235:109412.

Guo, C.Y., Xu, P., Wang, W., Han, Y., Kuai, Y.F. and Fan, Y.W., 2021b, "Experimental study on bubble sweep-down characteristics of research vessels", Ocean Engineering, Vol.228:108963.

Hino, T., Stern, F., Larsson, L., Visonneau, M., Hirata, N., and Kim, J., 2020, "Numerical Ship Hydrodynamics: An Assessment of the Tokyo 2015 Workshop", Springer International Publishing.

Hysa, I. , Tuinstra, M , Sciacchitano , A., and carano, F., 2023, "A multi-directional redundant 3D-PIV system for ship deck wind interactions", 15th International Symposium on Particle Image Velocimetry – ISPIV 2023, San Diego, California, USA.

IACS, 2022, "Guidelines on numerical calculations for the purpose of deriving the Vref in the framework of the EEXI regulation", IACS Rec., No. 173.

IMO, 2022a, "2022 guidelines on the method of calculation of the attained energy efficiency existing ship index (EEXI)", Annex 12: Resolution MEPC., 350(78).

IMO, 2022b, "2022 guidelines on survey and certification of the attained energy efficiency existing ship index (EEXI)", Annex 12: Resolution MEPC., 351(78).

ITTC, 1978, "Report of Performance Committee. In Proceedings of 15 ITTC", 15th International Towing Tank Conference, Hague, Netherlands, pp. 361-369.

ITTC – Recommended Procedures and Guidelines 7.5 – 03 – 02 – 02 Benchmark Database for CFD Validation for Resistance and Propulsion.

Jacobi, G. and Nila, A., 2021, “Simultaneous PIV and DIC Measurements in a Towing Tank Environment with a Flexible Hydrofoil”. In 14th International Symposium on Particle Image Velocimetry, Vol. 1, No. 1.

Jacobi, G., Thill, C.H. and Huijsmans, R.H.M., 2022, “Pressure reconstruction from PIV measurements in the bow region of a fast ship”, Ocean Engineering, Vol.252: 110318.

Jacobi, G., 2023, “Pressure reconstruction on a T-foil from underwater multi-plane stereo PIV measurements in a towing tank”, 15th International Symposium on Particle Image Velocimetry – ISPIV 2023. June 19–21, 2023, San Diego, California, USA.

JoRes Project, 2024, “Development of an industry recognised benchmark for Ship Energy Efficiency Solutions”, internet reference, https://jores.net/

Kanninen, P., P. Peltonen, V. Vuorinen, 2022, "Full-scale ship stern wave with the modelled and resolved turbulence including the hull roughness effect", Ocean Engineering, Vol.245: 110434.

Kerkvliet, M., Baltazar, J., Schuiling, B., Eça, L., 2024, “A numerical study on Model Propeller Performance Prediction Including Transitional and Passively Controlled Boundary Layer Considerations”, 8th Int. Symp. on Marine Propulsors, Berlin, Germany.

Kinnas, S.A., 2015, “Proceedings of the second workshop on cavitation and propeller performance”, The Fourth International Symposium on Marine Propulsors, SMP’15, Austin, Texas.

Korkmaz, K. B., Werner, S., Bensow, R., 2021, “Verification and validation of CFD based form factors as a combined CFD/EFD method”, Journal of Marine Science and Engineering, Vol.9(1), p.75.

Korkmaz, K., Kim, K., Liefvendahl, M., Werner, S., and Orych, M., 2023, “A Validation Study of Full-Scale CFD Simulation for Sea Trial Performance Prediction of Ships”, 10th Conference on Computational Methods in Marine Engineering, Madrid, Spain.

Kunihide, O., 2021, "Numerical study of roughness model effect including low Reynolds number model and wall function method at actual ship scale", Journal of Marine Science and Technology Vol.26, pp. 24-36.

Larsson, L., Stern, F., Visonneau, 2014, “Numerical Ship Hydrodynamics: An assessment of the Gothenburg 2010 Workshop”, Springer International Publishing.

Liefvendahl, M. and Finnsgard, C., 2024, “Ship power predictions with CFD in full scale”, SSPA/RISE Report, No: 2023:21, 2023.

M. Birvalski1, G. D. Struijk, D. Ponkratov., “Full-scale PIV measurements of the propeller inflow”, 15th International Symposium on Particle Image Velocimetry – ISPIV 2023, San Diego, California, USA.

Mikulec, M. and H. Piehl, 2023, "Verification and validation of CFD simulations with full-scale ship speed/power trial data", Brodogradnja, Vol.74(1), pp. 41-62.

Niklas, K. and H. Pruszko, 2019, "Full scale CFD seakeeping simulations for case study ship redesigned from V-shaped bulbous bow to X-bow hull form", Applied Ocean Research, Vol.89, pp. 188-201.

Ohashi, K., 2021, “Numerical study of roughness model effect including low Reynolds number model and wall function method at actual ship scale”, Journal of Marine Science and Technology, Vol.26, pp. 24-36.

Ortolani, F., Capone, A., Dubbioso, G., Pereira, F.A., Maiocchi, A. and Di Felice, F., 2020, “Propeller performance on a model ship in straight and steady drift motions from single blade loads and flow field measurements", Ocean Engineering, Vol.197: 106881.

Orych, M., Werner, S., and Larsson, L., 2021, "Validation of full-scale delivered power CFD simulations", Ocean Engineering, Vol.238: 109654.

Pena, B., E. Muk-Pavic, G. Thomas, P. Fitzsimmons, 2020, "An approach for the accurate investigation of full-scale ship boundary layers and wakes" Ocean Engineering, Vol.214: 107854.

Ponkratov, D., 2023, "JoRes Joint Research Project - the largest global community developing a benchmark for ship scale CFD", 25th Numerical Towing Tank Symposium, Portugal.

Ponkratov, D. and Wheeler, M., 2024, "Ship-Scale CFD Self-Propulsion validation – SMP'2024 vs SMP'2015", 8th International Symposium on Marine Propulsors, Berlin, Germany.

Ponkratov, D. and Struijk, G., 2023, "Quantifying the Uncertainty of High-Fidelity Speed/Power Trials", 8th Hull Performance and Insight conference HullPIC2023, Pontignano, Italy, pp. 28-30.

Quadvlieg, F.H.H.A., Flensborg-Otzen, L., Stern, F., Yasukawa, H., Kim, Y., Wentao, W., Yang, H. 2023a, "SIMMAN2020: Final Report before the workshop", Volume I; Captive conditions; Version 21 June 2023.

Quadvlieg, F.H.H.A., Flensborg-Otzen, L., Stern, F., Yasukawa, H., Kim, Y., Wentao, W., Yang, H. 2023b, "SIMMAN2020: Final Report before the workshop". Volume II; Trajectories; Version 16 June 2023.

Richardson, L. F., 1911, "IX. The approximate arithmetical solution by finite differences of physical problems involving differential equations, with an application to the stresses in a masonry dam", Philosophical Transactions of the Royal Society of London. Series A, Containing Papers of a Mathematical or Physical Character, 210(459-470), 307-357.

Roache, P., 1994, "Perspective: A Method for Uniform Reporting of Grid Refinement Studies," ASME J. Fluids Eng., Vol.116(3), pp. 405-413.

Roache, P. J., 1998, "Verification and Validation in Computational Science and Engineering", Albuquerque, NM: Hermosa, Vol. 895, p. 895.

Sakamoto, N., Kobayashi, H, Ohashi, K., Kawanami, Y., Winden, B. and Kamiirisa, H., 2020, "An overset RaNS prediction and validation of full scale stern wake for 1,600TEU container ship and 63,000 DWT bulk carrier with an energy saving device", Applied Ocean Research, Vol.105: 102417.

Savio, L., Jenssen, Y., Henry, P.Y. and Franzosi, G., 2024, "Experimental investigation on the flow and deformation fields of an elastic propeller". 8th International Symposium on Marine Propulsors, Norwegian University of Science and Technology, Department of Marine Technology, pp. 747-752.

Sanada, Y., Starman, Z., Bhushan, S. and Stern, F., 2023, "Four-dimensional particle tracking velocimetry measurements of unsteady three-dimensional vortex onset and progression for 5415 straight ahead, static drift, and pure sway", Physics of Fluids, 35(10).

Schouten, D. R., Drouet, A., Birvalski, M., and Morand, L., 2022, "Full Scale CFD Validation Using Ship Performance and Wave Pattern Measurements of a Mega Cruise Ship", American Society of Mechanical Engineers, Volume 7: CFD and FSI, page V007T08A041.

Sidi, A., 1997, "A Complete Convergence and Stability Theory for a Generalized Richardson Extrapolation Process," SIAM J. Numer. Anal., Vol. 34, No. 5, pp. 1761-1778.

Stern, F., Wilson, R. V., Coleman, H., and Paterson, E., 2001, "Comprehensive approach to verification and validation of CFD simulations – Part 1: Methodology and procedures," Journal of Fluids Engineering, Vol.123, pp. 793-802.

Stern, F., Diez, M., Satat-Hosseini, H., Yoon, H., and Quadvlieg, F., 2017, "Statistical Approach for Computational Fluid Dynamics State-of-the-Art Assessment: N-Version Verification and Validation," J. Verif. Valid. Uncert., 2(3): 031004.

Schouten, D. R., Drouet, A., Birvalski, M., and Morand, L., 2022, "Full Scale CFD Validation Using Ship Performance and Wave Pattern Measurements of a Mega Cruise Ship", International Conference on Offshore Mechanics and Arctic Engineering, ASME, Vol.85925.

Setiawan, H., Philip, J. and Monty, J.P., 2022, "Turbulence characteristics of the ship airwake with two different topside arrangements and inflow conditions", Ocean Engineering, Vol.260: 111931.

Seok, W., G. H. Kim, J. Seo, S. H. Rhee, 2019, "Application of the design of experiments and computational fluid dynamics to bow design improvement", Journal of Marine Science and Engineering, Vol.7(7), p. 226.

She, W., Zhou, G., Guo, C., Wu, T. and Song, K., 2021, "Experimental investigation on the water entry of a bulbous bow based on TR-PIV", Ocean Engineering, Vol.229, p. 108977.

Song, S., Y. K. Demirel, M. Atlar, S. Dai, S. Day, O. Turan, 2020, "Validation of the CFD approach for modelling roughness effect on ship resistance", Ocean Engineering, Vol.200: 107029.

Sun, C., Guo, C., Wang, C., Wang, L. and Lin, J., 2020, "Numerical and experimental study of flow field between the main hull and demi-hull of a trimaran", Journal of Marine Science and Engineering, Vol.8(12), p. 975.

Sun, W., Hu, Q., Hu, S., Su, J., Xu, J., Wei, J., Huang, G., 2020, "Numerical Analysis of Full-Scale Ship Self-Propulsion Performance with Direct Comparison to Statistical Sea Trail Results". Journal of Marine Science and Engineering, Vol.8(1):24.

Townsin RL, Medhurst JS, Hamlin NA and Sedat BS, 1984, "Progress in calculating the resistance of ships with homogeneous or distributed roughness", NECIES Centenary Conference in Marine Propulsion, Newcastle upon Tyne, United Kingdom.

Wilson, R., Stern, F., Coleman, H., and Paterson, E., 2001, "Comprehensive Approach to Verification and Validation of CFD Simulations-Part2: Application for RANS Simulation of a Cargo/Container Ship," ASME J. Fluids Eng., Vol. 123, Dec. 2001.

Wu, T., Luo, W., Jiang, D., Deng, R. and Li, Y., 2020, "Stereo particle image velocimetry measurements of the wake fields behind a Panamax bulker ship model under the ballast condition", Journal of Marine Science and Engineering, Vol.8(6), p.397.

Xing, T. and Stern, F., 2010, "Factors of safety for Richardson extrapolation", J. Fluids Eng. 132:6.

Xing, T. and Stern, T. 2015, Comments on "A procedure for the estimation of the numerical uncertainty of CFD calculations based on grid refinement studies", Journal of Computational Physics, Vol.301, pp. 484-486.

Yazaki, A., 1969, “A diagram to estimate the wake fraction for an actual ship from model tank test”, Proceedings of the 12th International Towing Tank Conference, Rome, Italy.

Zhu, N., Zhang, Z., Gnanamanickam, E.P. and Leishman, J.G., 2024, “Estimation of 3D ship airwakes using dual-plane PIV”, Experiments in Fluids, Vol.65(1), p.11.

Appendix A

The Combined EFD/CFD Specialist Committee proposes the following topics for future work.

The 7.5-03-01-01 guide has been substantially updated, and it is proposed to continue to promote its adoption, while gathering feedback about its use for its further refinement. There is a need to update other guides that rely on the 01-01 guide to reflect the changes in the 01-01 guide itself.

Currently there is not a guide for VV&UA for transient flows, and this is necessary for CFD to be used for seakeeping computations. It is proposed that in the future the 01-01 guide is expanded for transient flows, or that a new guide is developed for CFD in seakeeping.

The SC on Combined EFD/CFD in the 29th ITTC formed a list of issues affecting scaling and performance prediction. We reprint the list here and use it to make updated suggestions for future work. The list is ordered with the highest priority first:

1. Roughness allowance. Currently there are many researchers using CFD to understand roughness allowances. The next committee could gather the new information available and incorporate it into ITTC procedures and guides.

2. Finite-depth and restricted-water effects for full scale operation.

3. Propeller open water scaling (transition modelling at model scale). This work should consider both conventional and unconventional propellers, including azimuthing thrusters.

4. Flow separation and vortices shed the vessel

5. Energy saving devices

6. Transom drag

7. Wave resistance

8. Appendage resistance

9. Hull friction estimation using alternative friction or correlation line

Work that has already been initialized, but can benefit from further attention:

10. Nominal and effective wake scaling. This topic was studied by the SC on EFD/CFD in the 30th ITTC, and while progress was made towards understanding scaling of wakes, this topic could be continued in the future to develop a new technique to determine the full-scale wake fraction.

11. Form factor determination. This topic was studied by the SC on EFD/CFD in the 29th ITTC.