Best Practice Guidelines on the *Ansys* in Building Wind Action Applications

GUILHERME S. TEIXEIRA, MARCO D. DE CAMPOS
Institute of Exact and Earth Sciences,
Federal University of Mato Grosso,
Av. Valdon Varjão, 6390, Barra do Garças, 78605-091, Mato Grosso,
BRAZIL

Abstract: - In the last 30 years, *Computational Fluid Dynamics* use in *Wind Engineering* has allowed researchers to raise its capabilities, and, little by little, it is becoming a reasonable tool because of the availability of high-performance computers with large storage capacities. This work offers a short guide to the twelve tips computer beginners can take on the *Ansys* in building wind action applications.

Key-Words: - Best practice, guideline, Wind Engineering, A*NSYS*, computer beginners, CFD, numerical simulation.

Received: November 15, 2022. Revised: August 25, 2023. Accepted: October 4, 2023. Published: November 1, 2023.

1 Never Use Special Characters or Spaces in File Naming

In recent decades, new ways of analyzing and designing structures for wind loads have emerged due to progress in information technology, data storage methods, and computational tools.

Among these last, *Ansys Workbench* (often *ANSYS WB*) established itself among researchers because of its integrity and cost-effectiveness. Next, twelve best practice guidelines on the *Ansys* in building wind action applications will be presented and discussed.

Possibly, this is the most common mistake made by beginners in simulations or programming in general. A computer scientist could use an entire article to explain why special characters become a problem in file naming, [1]. In the scope of a beginner user in *Ansys WB*, it is enough to say that some operating systems can assign specific functions to certain characters as commas, quotes, ampersands, and parentheses, among others. Consequently, its use in the simulation file directory can generate incompatibilities, compromising file accessibility or even causing it to be deleted or inaccessible. Choosing inappropriate file names or addresses can result in two implications. The first is about the failure of the simulation in *Ansys WB* due to its not finding the file directory. The second is the inoperability between the operating systems since it is possible to use *Ansys* collaboratively, sharing the files of the simulation steps (Figure 1) with other computers. In the context of a group of researchers, it is natural to use equipment with different operating systems, and it may happen that a particular character accepted in one does not work in another (as is the case with *Windows* and *Linux*, for example). This way, by simply renaming the address using only non-special characters, this error could be eliminated, as well as underlines and hyphens instead of spaces. It is recommended from the beginning.

2 Appropriately Model your Control Volume

In an Eulerian approach, a control volume (CV) is the region of space through which the fluid flows in and out, [2]. However, this basic definition in *Computational Fluid Dynamics* (CFD) may be ambiguous for beginners when used in a project to create a fluid domain in *Ansys WB*. This fact occurs because, according to the definition, in the simulation domain definition, it would be enough to model only the region through which the fluid flows. Thus, it would be necessary to model only the surfaces obstructing the flow, that is, as a unique solid body (Figure 2). Thus, the control volume must be created by extracting the geometry of interest from its interior, causing only one obstruction in the flow. In other words, there is no need to insert the extracted geometry again, which would result in two elements in the simulation.

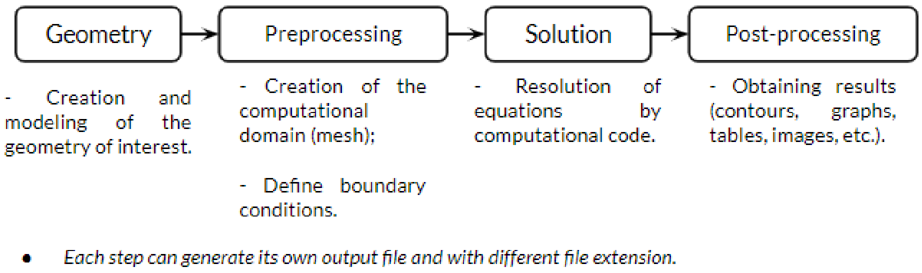


Fig. 1: Computational simulation steps.

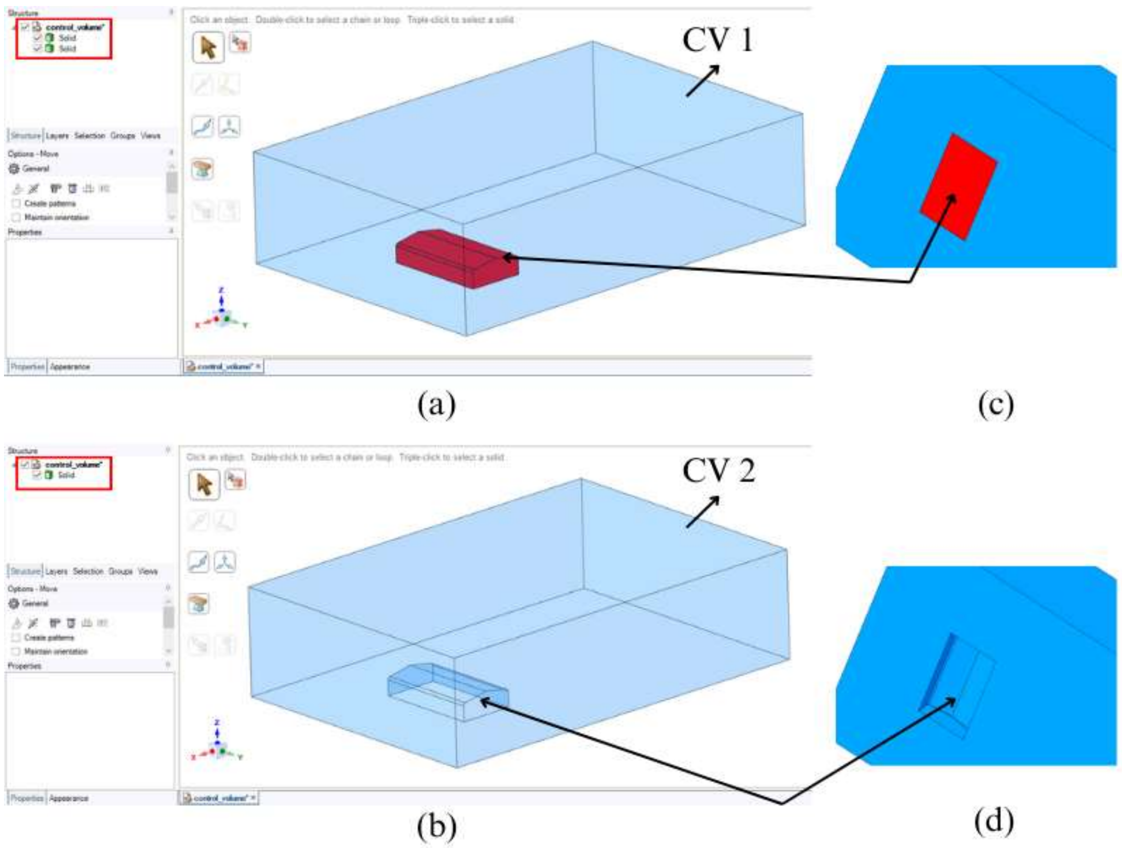


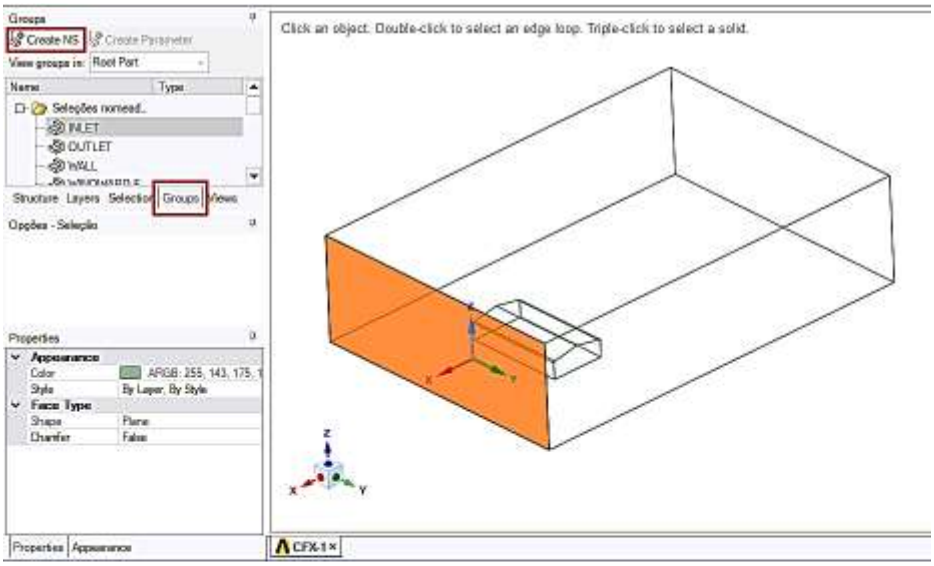
Fig. 2: Screenshot of *SpaceClaim* (*Ansys WB CAD* platform) showing (a) an inadequate CV, with two solid bodies in the model, one blue (CV) and one red (building), (b) an adequate CV (note that there is only one body with an obstruction of the same volume and shape as the building of interest), (c) obstruction filled in the inappropriate CV (CV 1, top view) and, finally, (d) existing hollow obstruction in the appropriate CV (CV 2, bottom view).

3 Make Use of Software Features

A good directive to produce an adequate simulation is to know how to use the software tools. In the preparation of the geometry, the "*Named selections*" in *Ansys WB* allows the user to name the faces or a set of them (i.e., INLET, OUTLET, WALL, WINDWARD ROOF, LEEWARD ROOF, that is, whatever the user wants) (Fig. 3-a).

Naming the faces according to the lists of each component of the analysis system (for example, geometry mesh, or setup) becomes useful for creating boundary conditions and plotting the results. On the other hand, unnamed faces showed in the form of "*codes*" (like F106.136, F107.136, F108.36) (Fig. 3-b).

Thus, initially, it is recommended to name the most important faces of the model in the component *"geometry"*. Thus, the time in building the simulation will be optimized, in addition, to minimizing errors in the boundary conditions.



(a)

(b)

Fig. 3: Screenshot showing (a) the “Named Selection” tool that is accessed by clicking with the mouse button on the face to be named and (b) a typical list of components of an *Ansys WB* analysis component (named and unnamed faces).

4 Prepare your Geometry According to Available Post-Processing

Improving the quality of the simulation output data is directly related to the computational model development according to the expected results. This fact means not only expecting a physically coherent result due to the physical behavior of the fluid and its possible effects on the structure but also how these results will be presented and treated Beforehand, it is necessary to determine which part of the structure (entirely or in part) the variables will be calculated (*Cpe*, streamlines, or velocity vectors, for example). This consideration extends to streamlines, velocity vectors, or any other parameter typical of Wind Engineering. This treatment is called *post-processing*.

Suppose a hypothetical case in which the user selects the four facades of a shed and names them "FACADES" (using the "Named selections" tool). In this way, it will be not possible to plot the contours of *Cpe* only on the front and rear façade, for example. In plotting the results, the FACADE must be considered integrally (the four facades together). However, identifying the four facades separately can be plotted the results individually or together.

In addition, other combinations will be possible: the user could present the contour maps on the four facades, only on the front facades and, later, together or only on the leeward roof, in case the windward is not the object of study, for example. When plotting the results for some specific parts of the geometry, it is necessary to prepare the model in advance. Thus, when *Cpe* is graphically represented only on the roofs, excluding the facades, it is required to name them separately. This methodology results in greater versatility in the acquisition and analysis of results of spaces.

DOI: 10.37394/232013.2023.18.12

5 Build Meshes According to your Processing Power

Although the RAM capacity of the hardware is a less critical factor than the machine's processor, it is still a point that deserves attention. Defining the maximum number of mesh elements to be processed as a function of the RAM size is a hard decision. The reason is that this relationship is not direct and depends, among other factors, on the mesh, the solution algorithm, numerical precision, and connectivity between elements. In the case of hardware from large research groups (in academia or industry) or even professionals in the area who work with many different simulation cases, the ideal is to benchmark the machines to establish some parameters of the RAM-Mesh ratio.

However, in the case of wind engineering beginners using *Ansys*, some recommendations may be convenient to understand the particularities of this type of simulation. Although some authors, [3], recommend a ratio of 1GB for every 1 million mesh elements, the indication of 2GB for every 1 million cells is a *golden rule* in many forums and theme pages. For this reason, simulation software developers do not indicate the use of machines with less than 4GB RAM since this is the minimum value by *Ansys WB*.

In this context, the most important thing for the beginner would be to understand the coherence between the mesh built for the computational model and the machine available to solve such a problem. Initially, you should test values in the range of 1 to 2GB for every million cells and, later, with experience, be able to establish and analyze the capacity of the hardware itself.

6 Build Meshes According to your License Limitations

When using limited licenses, attention is necessary for the model mesh construction. For the student version, for example, the mesh limitation is 512000 nodes for fluid simulation (for the latest version at the time of writing this article).

Ansys Workbench allows the generation of refined meshes as supported by the hardware. However, the limitation will appear in the solver (*CFX* or *Fluent*).

When exceeding the software limit, when solving the computational model, a typical message will appear (Fig. 4-a), and by not explaining the problem, it may confuse the novice user. In this case, to understand the error, it is necessary to check the simulation out file (Fig. 4-b). For this reason, before proceeding to the equation solution step, it is recommended to verify the number of elements and nodes created in the mesh step, paying particular attention to the license limitation, if any.

Fig. 4: (a) Screenshot of typical message for several troubleshooting errors in *Ansys Workbench*, including extrapolation of several cells/nodes limited by license and (b) warning, in the out file, specifying the error.

7 Equation Discretization Schemes

CFD consists of solving equations through computer codes, [2], including *Computational Wind Engineering* (CWE), [4]. However, the physical nature of fluids governed by such equations is not trivial, leading to complex mathematics. Therefore, an appropriate methodology, including discretization, is necessary to make the governing equations of Fluid Mechanics solvable by computer codes. Thus, these differential equations must be rewritten as simpler algebraic equations. These equations have different terms (e.g., temporal term, advective term, diffusive term, and source term), and, in *Ansys WB*, it is possible to choose the numerical approximation for the advective terms and the additional turbulence equations. For that, there are two options: Upwind and High Resolution (Fig. 5). The first is called first-order discretization and produces less accurate results than the second; however, its computational cost is also lower. Upwind discretizations provide the preliminary simulation results. However, the literature recommends that for the final results of your model, second-order discretizations, called High Resolution by *Ansys WB*, be used, [5].

In addition to the computational cost and accuracy of the results, it is worth mentioning that the acceptance of the research is also subject to this boundary condition. Some journals do not accept simulations whose results do not use high-order discretization schemes, [6]. Therefore, if possible, use second-order discretization schemes for your final results.

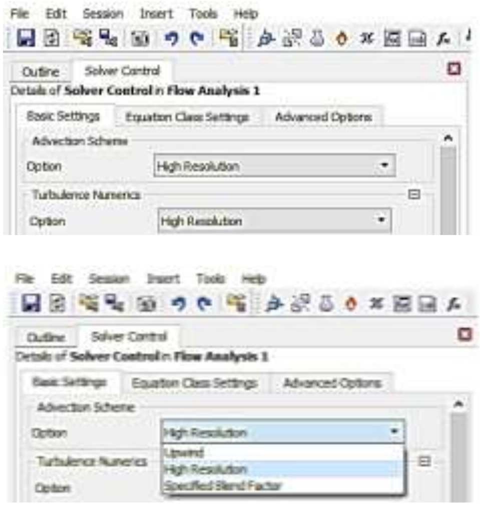


Fig. 5: Screenshot of the equation discretization scheme menu using the *Ansys Workbench* setup component.

8 Unexpected Interruption of the Simulation

A common problem in commercial simulation software occurs when the solution processing stops unexpectedly. In *Ansys WB*, with the "solution" component started, care must be taken that the software is not closed during this process. If this happens, due to carelessness on the part of the user, or even a power outage, or the end of the battery, the simulation will then be interrupted and, when returning to the *Ansys WB* initial screen, a warning will inform about the end of the processing (Fig. 6).

The user will have to choose between recovering the files and data created up to the moment of interruption or abandoning them, returning with the data from the last savepoint before the stage of the simulation starts.

In this case, the recommendation is to discard the data generated up to the moment of interruption, considering only those already saved. In broken data recovery, the file may have become corrupted.

In this context, it is also relevant to encourage beginners in simulation to keep their files updated periodically during the construction of the computational model. Also, avoid, when starting a file, saving it only after post-processing or even after carrying out a large volume of processing.

Fig. 6: Screenshot of the message when restarting the file after stopping a numerical solution.

9 Vertical Profiles

Ansys WB, like other software, is based on a Cartesian coordinate system (x,y,z). When using a vertical wind profile, such as the *Power Law* (function of a z height), the control volume placement must be correct to avoid physical inconsistencies in the results. For example, when building the control volume with the terrain positioned at a height z<0, consequently, the user will obtain negative velocities in the input profile. In this case, a negative velocity vector on the input face (INLET, i.e., the facade through which the fluid enters and does not leave) will result in a vector in the opposite direction to the input plane, which is considered an inconsistency in the model (Fig 7-b). This way, there will be an error, and the simulation will not be complete. Thus, for vertical wind profiles modeled from Power Law or Logarithmic Law, when building its geometry, it is necessary to ensure that the lowest terrain plane (i.e., the face that represents the terrain in the computational model) is positioned at z *height* = 0 (Fig 7-a). This recommendation applies to the most common cases of simulation of wind action in buildings, such as control volumes that include open and flat land, flat urban areas, among others. It is worth mentioning that the use of the *absolute function abs( )* to prevent this possible error ends up generating another one related to the inadequate form of the *Power* or *Logarithmic Law profile* (Fig 7-c).

10 A Warning is not an Error (But it May be an Indication to Review your Choices)

When executing some commands, *Ansys* may issue alert notices. For example, when inappropriately choosing a structured mesh model made up of parallelepipeds for a geometry that would perform better with an unstructured mesh. In this case, the user will receive the message that some elements initially foreseen were not created (Fig. 8).

Sometimes, this amount can be an indication to review the choice of mesh method or use another refinement in the geometry. In addition, when defining an OUTLET-type boundary condition on a face and verifying a reverse flow during the resolution of the equation, the software will issue a warning indicating the change to OPENING, which allows reverse flow.

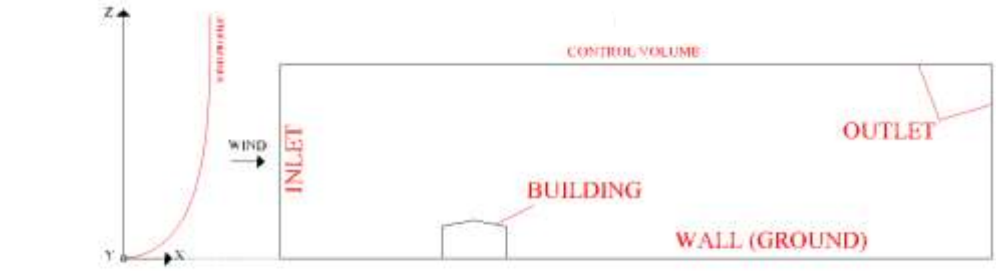
Fluid re-entry through the OUTLET face is defined as *overflow* by *Ansys WB*. So, pay attention to the messages provided by the software and rethink whether the chosen model choices are adequate.

11 Choose an Appropriate Turbulence Model

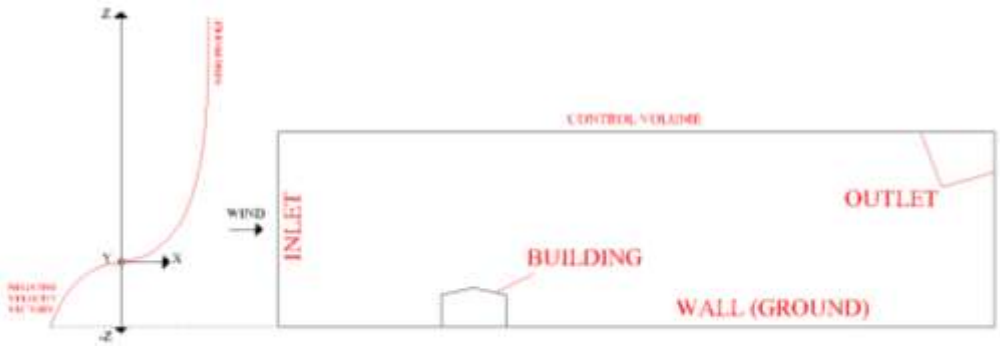
In *Computational Wind Engineering* (CWE), the choice of the turbulence model is one of the most relevant points for a successful simulation. This boundary condition can vary with the type of problem, whether pedestrian-level wind environment, near-field pollutant dispersion, natural and urban ventilation, structural wind engineering, or others. Each of these problems requires specific computational modeling for turbulence.

In addition, commercial software such as *Ansys WB* has some options available (Fig. 9). In this context, some researchers have investigated which models perform better for different simulation cases, [7], [8], [9].

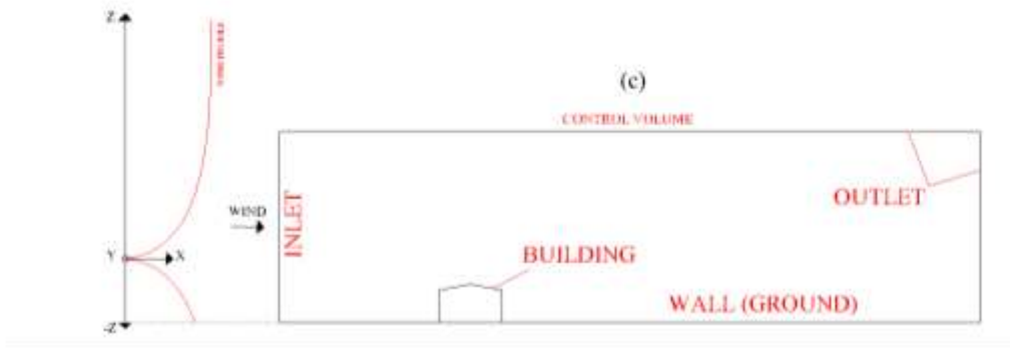
For the simulation of wind action on buildings, some models are widely used as low and high-rise buildings, industrial buildings, and residential buildings, among others. Generally, two main groups, with their variations, are employed: *k-ε* and *k-ω models,* [10]. For the *k-ε models*, the main variations applied are the *Standard k-ε (Skε),* [11], *Realizable k-ε* (*Rkε*), [12], *Renormalization Group k-ε* (*RNG k-ε*), [13]. The *k-ε RNG* has better indications in the literature than the *Skε* and is usually more used in bluff bodies, [14]. In the *k-ω group*, the *Shear Stress Transport k-ω,* [15], is the most used. When preparing a simulation, one should look for references of turbulence model applications applied to cases similar to the one of interest. This practice is known as *benchmarking*.



(a)



(b)



(c)

Fig. 7: (a) Appropriate and (b) inadequate positioning of the control volume in the Cartesian axes system, and (c) the physically inconsistent profile generated by using the *abs( ) function*.

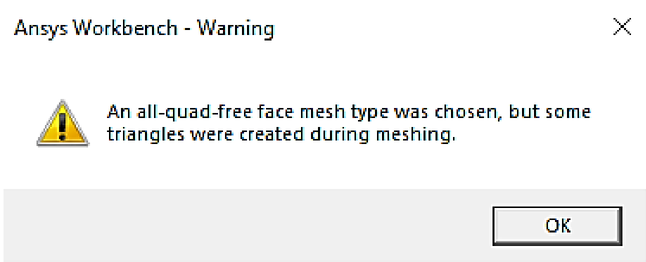


Fig. 8: Warning about problems with mesh generation.

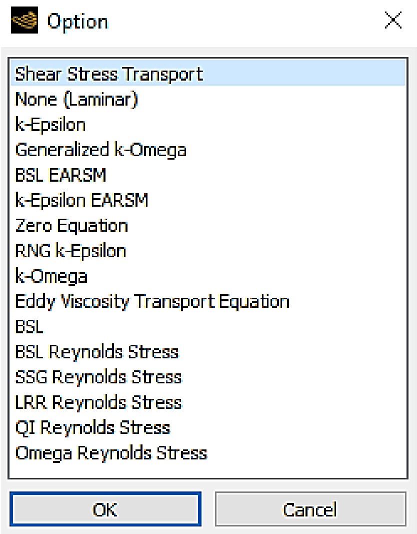


Fig. 9: Turbulence models are available on *Ansys WB*.

12 Best Practices Guidelines Known in The Literature

Other researchers have listed relevant recommendations in *Computational Fluids Dynamics* (CFD). The present text is a contribution focusing on the commercial software *Ansys WB*. In this context, these texts will allow a basis for the best choices regarding the computational model. For this reason, we list some authors and their most famous guidelines, [5], [16], [17], [18], [19], [20], [21]. In particular, when it comes to *Ansys WB*, it is recommended to consult the guides by *Ansys* Inc. They are technical texts in which it is possible to select the topics of interest, [22], [23], [24], [25].

13 Conclusions

Ansys WB is one, among others, a tool that addresses many engineering problems. This work presented some good practices related to geometry, meshes, boundary conditions preparations, and common software-specific errors.

These guidelines resulted from the authors' personal experience with *Ansys WB* and a software-oriented view of good practices of literature.
In summary, we present the following checklist with topics and questions that beginners of simulation with *Ansys WB* should pay attention to, namely:

Step 1: Check the file's email address
a. Ensure that the address is free of special characters or empty spaces;
b. Use only letters, numbers, underscores, and hyphens.

Step 2: Check that the geometry is optimized (any region of space through which the fluid does not flow is unnecessary in the computational model.)

Step 3: List your preliminary decisions
a. What variable types do you want to get?
b. How will it be analyzed and presented?
(The answers to these questions will help you build a targeted model for your CFD solution).

Step 4: Analyze your mesh choice.
a. Check if the mesh type is best suited to your geometry.
b. Also observe whether the number of elements and nodes in your mesh are compatible with your computing power and software license (if any).

Step 5: Understand your boundary conditions.
a. Know the physics involved in the problem;
b. Check the properties of the fluid (density, viscosity, temperature);
c. Also check the input variables (such as pressure) and the mathematical expressions for the speed profiles.

Step 6: Dealing with errors.
a. Check the error description.
b. Then consult the software Help Center.
c. If necessary, check forums on the developer company's website and independent forums.

This script, associated with the tips detailed throughout the text, is a manual for beginners in wind simulation with *Ansys WB*.

Future work may add suggestions for good practices aimed at Wind Engineering with *Ansys WB* in addition to those explained here and apply them to other CFD software.

References:

[1] M. F. O. Rosenbloom and A. L. and Carpenter, Macro Quoting to the Rescue: Passing Special Characters. In: *Proceedings of SAS Global Forum,* 2013.

[2] Y. A. Çengel and J .M. Cimbala, *Fluid Mechanics*: *Fundamentals and Applications*. New York: McGraw-Hill, 2014.

[3] ANSYS, *ANSYS Mechanical APDL Performance Guide*, Canonsburg, 2013.

[4] M. Thordal, J. C. Bennetsen and H. H. H. Koss, Review for practical application of CFD for the determination of wind load on highrise buildings, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 186, 2019, pp. 155-168.

[5] J. Franke, C. Hirsch, A. G. Jensen, H. W. Krüs, M. Schatzmann, P. S. Westbury, S. D. Miles, J. A. Wisse and N. G. Wright, *Recommendations on the use of CFD in predicting pedestrian wind environment, COST Action C14: Impact of Wind and Storms on City Life and Built Environment. Hamburg*, COST Office, 2004.

[6] C. J. Freitas, Editorial policy statement on the control of numerical accuracy, *Journal of Fluids Engineering*, Vol. 115, No. 3, 1993, pp. 339-440.

[7] B. Lia, J. Liua, F. Luoa and X. Man. Evaluation of CFD Simulation using various turbulence models for wind pressure on buildings based on wind tunnel. *Procedia Engineering*, Vol. 121, 2015, pp. 2209-2216.

[8] G. K. Ntinas, X. Shen, Y. Wang. Evaluation of CFD turbulence models for simulating external airflow around varied building roof with wind tunnel experiment. *Building Simulation*, Vol. 11, 2018, pp.115–123.

[9] M. Xiong, B. Chen, H. Zhang and Y. Qian. Study on Accuracy of CFD Simulations of Wind Environment around High-Rise Buildings: A Comparative Study of k-ε Turbulence Models Based on Polyhedral Meshes and Wind Tunnel Experiments. *Applied Sciences*, Vol. 12, No. 14, 2022, 12, paper 7105.

[10] J. Franke, C. Hirsch, A. G. Jensen, H. W. Krüs, M. Schatzmann, P. S. Westbury, S. D. Miles, J. A. Wisse , N. G. Wright. Recommendations on the use of CFD in wind engineering. In: *Proceedings of the International Conference on Urban Wind Engineering and Building Aerodynamics*, 2004.

[11] B. E. Launder, D. B. Spalding, *Mathematical Models of Turbulence*. New York: Academic Press, 1972.

[12] T. H. Shih, W. W. Liou, A. Shabbir, Z. Yang, J. Zhu, J., 1995. A new k-ε eddy viscosity model for high Reynolds Number turbulent flows-model development and validation. *Computational Fluids*, Vol. 24, No. 3, 1995, pp. 227-238.

[13] V. Yakhot, S. A. Orszag, S. Thangam, T. B. Gatski, C. G. Speziale,. Development of turbulence models for shear flows by a double expansion technique. *Physics of Fluids* A4, 1992, pp.1510-1520.

[14] T. Potsis, Y. Tominaga, T. Stathopoulus. Computational wind engineering: 30 years of research progress in building structures and environment. *Journal of Wind Engineering & Industrial Aerodynamics*, Vol. 234, 2023, paper 105346.

[15] F. R. Menter, Eddy viscosity transport equations and their relation to the k-ε model. *Journal of Fluids Engineering,* Vol. 119, 1997, pp. 876-884.

[16] J. Franke, A. Hellsten, H. Schlünzen and B. Carissimo, *Best practice guide for the CFD simulation of flows in the urban environment, COST Action 732: Quality assurance and improvement of microscale meteorological models.* Hamburg: COST Office, 2007.

[17] J. Franke, A. Hellsten, H. Schlünzen and B. Carissimo, The COST 732 Best practice guideline for CFD simulation of flows in the urban environment: a summary. *International Journal of Environment and Pollution*, Vol. 44, 2011, pp. 419-427.

[18] T. Tamura, H. Kawai, S. Kawamoto, K. Nozawa, T. Ohkuma. Numerical prediction of wind loading on buildings and structures — activities of AIJ cooperative project on CFD. *Journal of Wind Engineering and Industrial Aerodymics*, Vol. 67–68, 1997, pp. 671-685.

[19] T. Tamura, K. Nozawa, K. Kondouma. AIJ guide for numerical prediction of wind loads on buildings. *Journal of Wind Engineering and Industrial Aerodymics,* Vol. 96, 2008, pp.1974-1984.

[20] Y. Tominaga, A. Mochida, R. Yoshie, H. Kataoke, T. Nozu, M. Yoshikawa and T.Shirasawa, AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 96, No. 10-11, 2008, pp. 1749-1761.

[21] Architectural Institute of Japan, 2016. *AIJ benchmarks for validation of CFD simulations applied to pedestrian wind environment around buildings.* Architectural Institute of Japan, 2016.

[22] ANSYS, *CFX-Solver Modeling Guide Ansys*, Canonsburg, 2009, [Online], https://dl.cfdexperts.net/cfd_resources/Ansys_Documentation/CFX/Ansys_CFX-Solver_Modeling_Guide.pdf (Accessed Date: August 22, 2023).

[23] ANSYS, *CFX-Solver Theory Guide Ansys*, Canonsburg, 2009, [Online], https://dl.cfdexperts.net/cfd_resources/Ansys_Documentation/CFX/Ansys_CFX-Solver_Theory_Guide.pdf (Accessed Date: August 22, 2023).

[24] ANSYS, *Meshing User's Guide Ansys*, Canonsburg, 2010, [Online], https://dl.cfdexperts.net/cfd_resources/Ansys_Documentation/Ansys_Meshing/Ansys_Meshing_Users_Guide.pdf (Accessed Date: August 23, 2023).

[25] ANSYS, *CFX-Solver Manager User's Guide Ansys*, Canonsburg, 2016, [Online], https://www.cfdlectures.com/tutorials/cfxtutorial.pdf (Accessed Date: August 22, 2023).

Contribution of Individual Authors to the Creation of a Scientific Article (Ghostwriting Policy)

Guilherme Teixeira was responsible for the methodology. Marco Campos carried out the conceptualization, review, and editing.

Sources of Funding for Research Presented in a Scientific Article or Scientific Article Itself

No funding was received for conducting this study.

Conflict of Interest

The authors have no conflicts of interest to declare.