AeroSim

Technical and Institutional Presentation

CNPJ: 41.534.456-0001/19

_____
Eng. Aron Letchacovski Zavelinski

Head of Engineering

CREA/PR 226481/D

aron@aerosim.io

_____
Waine Oliveira Junior

CEO

waine@aerosim.io

Alameda Cabral, 475 - São Francisco, Curitiba - PR, 80410-210

consultoria@aerosim.io

aerosim.io

SUMMARY

AeroSim

AeroSim was founded in 2021 by Dr. Alan Lugarini and computer engineer Waine Oliveira Jr., following both of their experience in R&D involving computational fluid dynamics (CFD) projects for the oil and gas industry. Since then, it has acquired expertise in conducting advanced wind studies for structures, having served more than 70 projects, not being limited to the Brazilian national market, but also to constructions in Latin America, Europe, and Asia.

It has also developed partnerships with major design offices, construction companies, and structural engineers of high-rise buildings and warehouses, with emphasis on Vendramini Engenharia, Precisão Estrutural, Estrutural Engenharia, Codeme, the structural engineer Sérgio Stolovas, and the Italian structural engineer Andrea Imbrenda.

Such work was only possible due to the development of proprietary computational fluid dynamics software focused on wind engineering, which demonstrated accuracy and performance far superior to market alternatives, as evidenced by the extensive validation portfolio, with more than 90 wind tunnel comparisons. Based on this, a series of guidelines for simulations and quality parameters were developed, which are constantly improved by the AeroSim team.

Currently, the software is made available to academic partners, with the quality of the software and our guidelines also being independently demonstrated.

Associations

We are associated with ABECE and are part of the Nvidia Inception program. Since 2021, incubated at Incubadora CELTA, of Fundação CERTI, an institution focused on innovation and technological development, promoting solutions for industry and society. In addition, it is a resident member of Accelera Hub, an innovation hub located in Curitiba focused on construction and urbanism, bringing together startups, construction companies, developers, suppliers, and investors, fostering strategic connections for the business.

Partnership with laboratories and research institutions is a key point in the company's strategy for technology validation and collaborative development. In Brazil, it collaborates with IPT and UTFPR, with access to the computational fluid dynamics laboratory through CERNN (Center for Research in Rheology and Non-Newtonian Fluids). Internationally, it has

partnerships with Nova Fluid Mechanics, the University of Bristol, and Concordia University, which in addition to collaborating in research, provide infrastructure with physical wind tunnels for experimental validations.

The company has participated in different public calls and innovation support programs and obtained funding to support its development, such as Sebrae Catalisa ICT, Centelha II – Fundação Araucária, Oracle for Startups, Nvidia Inception Program, and Finep Inovacred.

Technical Team

AeroSim has an experienced and multidisciplinary technical team, bringing together the essential competencies for business development. Currently the team is led by Aron Letchacovski Zavelinski, mechanical engineer graduated from UTFPR, with more than eight years of experience in computational fluid dynamics (CFD), working both in academia and professionally. Since 2021, he has directed his work toward wind engineering, developing methodologies for the use of CFD in wind studies applied to structures such as buildings and warehouses, with special focus on wind loads. He is currently Engineering Director at AeroSim and a master's student in the Graduate Program in Mechanical and Materials Engineering (PPGEM-CT), UTFPR, with research focused on wind engineering. His Lattes curriculum can be accessed at: lattes.cnpq.br/2743293930062251.

Services and Clients

The company has been operating since 2021 providing information that qualifies decision-making related to the effects of wind on civil construction projects. Through the application of high-fidelity transient computational fluid dynamics methods, we are able to demonstrate how the combination of terrain, buildings, and wind affects the main project parameters.

Our performance throughout this trajectory has been broad in scope, involving services such as estimation of topographic effects, external and internal pressures, forces on selected elements, dynamic effects, and pedestrian comfort assessment. We also operate in multiple Architecture, Engineering, and Construction markets, including the design of logistics warehouses, industrial buildings, hangars, ports, and buildings.

In our projects we have already worked with some of the largest real estate asset managers, developers, project managers, construction companies, and suppliers, such as: Brookfield, Cy.capital, GLP, Prologis, Golgi, Fullwood, Moura Dubeux, Monto, Libercon, JLC, Ribeiro Caram, Codeme, among others. We also operate with some of the main structural designers in the country, such as Estrutural Engenharia, Vendramini Engenharia, Precisão Estrutural, AS Estruturas, Kalkulo Projetos Estruturais, among others.

In addition, we have had projects with international partnerships, such as with Tian Building Engineering, a Singapore-based company specialized in computational studies for structures, Proges Engineering, an Italian structural consulting office, and Grupo Petra, a Paraguayan construction and development company.

Academic Partnership

Since its foundation, the company has sought to build a solid network of technical-scientific and commercial partnerships.

In 2025, we had work developed by a partner from the University of Bristol, using our technology, presented at the 15th International ERCOFTAC Symposium on Engineering Turbulence Modelling and Measurements, in Croatia.

Also noteworthy is the collaboration with TU Dortmund University, in a benchmark study with simulation data provided by our team and currently under review for publication, in addition to contacts with the University of La Coruña, in Spain, with interest in future research on transient flows in buildings. We have also advanced with Concordia University, in Canada, with the possibility of using its wind tunnel for experimental validations, expanding our academic partnership network.

In Brazil, we consolidated partnership with CERNN (UTFPR), which has provided access to research infrastructure and technical support, resulting in joint publications at events such as ICWE. Another milestone is the collaboration with CRIACIV, from Italy, one of the main wind engineering centers in Europe, with whom we have already conducted blind validation tests. In addition, AeroSim is an associated company of ABECE (Brazilian Association of Structural Engineering and Consulting) and participated in the most recent committees for revision of standard NBR 6123.

Academic publications of AeroSim members are presented in Annex B.

Use of CFD in Wind Engineering

AeroSim's positioning regarding the use of computational methods for determining wind loads and their dynamic effects is in line with the consensus of the academic community and with the state of the art. Our view is guided by the following principles:

- Physical wind tunnel testing remains the most accurate and reliable method for determining aerodynamic loads, with the sole exception of field measurements, which by their nature do not apply to structures in the design phase. Therefore, for highly complex projects, traditionally subject to mandatory wind tunnel testing, we do not consider CFD as a substitute or viable competitor, considering the current level of maturity of the technology.
- For projects in which aerodynamic design would be carried out exclusively by analytical methods, the use of CFD can represent a substantial advancement, by reducing uncertainties and offering a high level of detail regarding pressure fields, flow patterns, and load combinations.
- Numerical simulation (CFD) can be employed in conceptual phases of the project, contributing to accelerate the development cycle and reduce the number of wind tunnel iterations, optimizing the decision-making process.
- We do not endorse the unrestricted or unqualified use of CFD for executive design purposes, nor do we support the inclusion of this type of positioning in technical standards. On the contrary, we advocate the requirement of the use of high-fidelity transient models, with explicit validation of the procedures used, through comparison with representative experimental data - originating from geometries and conditions analogous to those of the project under analysis..
- We also encourage the inclusion in technical standards of well-defined performance criteria, which every method for determining wind loads - whether experimental or numerical - must be able to reproduce, ensuring traceability, reliability, and coherence in the results used for structural design purposes.

Annex A presents a bibliographic review of academic and normative positioning worldwide, which we use as a basis for our positioning.

Software

The technology developed by AeroSim involves a high degree of technical complexity, both in physical modeling and in its computational architecture. It is a solution based on the Lattice Boltzmann Method (LBM), an alternative approach in computational fluid dynamics (CFD), grounded in the Boltzmann equation of the kinetic theory of gases.
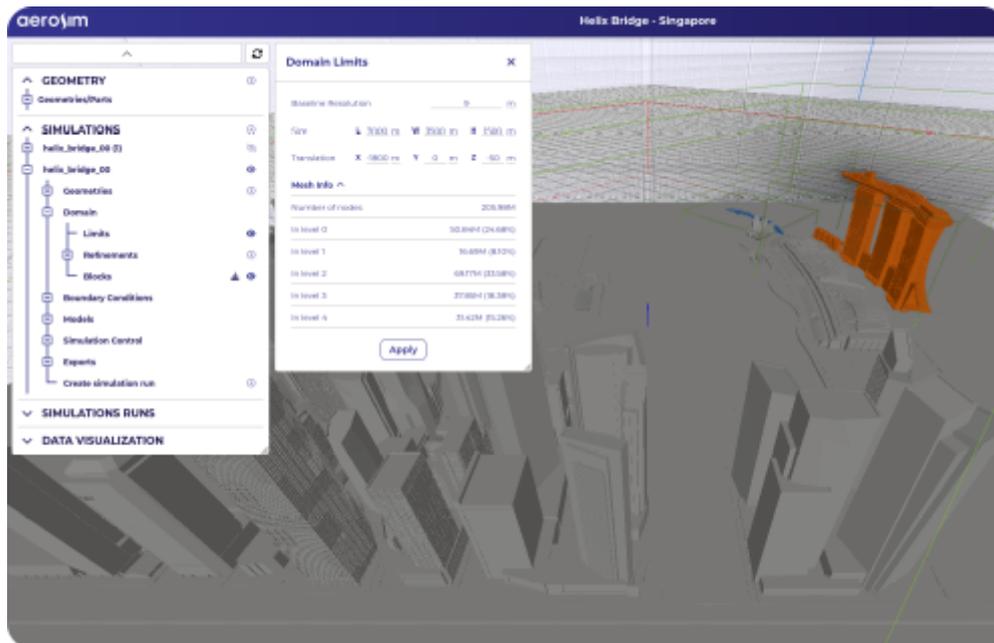
Unlike classical formulations based on the Navier-Stokes equations, which describe flow deterministically at the macroscopic scale, LBM adopts a statistical perspective, representing fluid behavior through the evolution of particle distributions at the mesoscopic scale. Although the LBM method was originally developed in the late 1980s, its use has only become more widespread recently, driven by advances in processing capacity and memory of graphics processing units (GPUs).

One of the main attractions of LBM is its algorithmic structure in which most operations can be performed locally, that is, without the need for communication between adjacent nodes. This characteristic makes LBM especially suitable for parallel execution on GPUs. This naturally parallel profile has renewed interest in optimizing LBM for high-performance environments, particularly in the context of scientific and industrial applications.

The computational differential of the software is significant: the current solution allows the simulation of a typical case in approximately 12 hours,

while conventional industry tools (based on LES) take approximately 5 days to deliver equivalent results. This performance is only possible thanks to technical expertise that combines understanding of the physics involved with advanced skills in parallelization and GPU programming. This set of requirements makes the challenge of materializing the solution particularly complex.
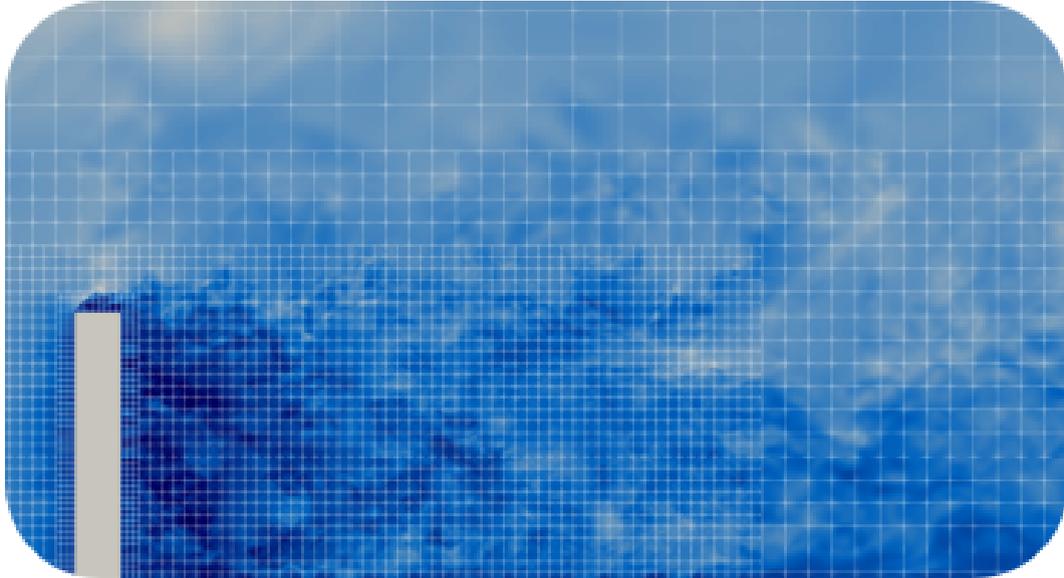
However, the consolidated experience of the executing team, combined with already established strategic partnerships and the future integration of key professionals, are determining factors to enable the successful execution of the project. The team's trajectory reflects a solid technical foundation, built over years of activity in applied research and development of computational solutions. From physical modeling of complex phenomena to the implementation of high-performance codes, the members have accumulated experience ranging from academia to industry. This background was decisive for the company to recognize the feasibility of creating a national high-efficiency software, exploring the Lattice Boltzmann Method (LBM) for practical applications in wind engineering.

Validations

Any method or tool that proposes to inform engineering decisions must necessarily justify its epistemic bases; in the context of numerical methods, this requirement corresponds to the classical framework of verification (mathematical and numerical correctness of the implementation) and validation (demonstration that the model reproduces observable physical phenomena within the domain of application). In wind engineering, the physical core is given by the Navier–Stokes equations, whose validity as a model of atmospheric flows is widely established, unlike strongly multiphysics contexts highly dependent on interdependent empirical constitutive laws. Thus, the epistemic focus shifts to the ability of the computational method to reproduce, with statistical accuracy, the quantities relevant to structural design: transient pressure and force fields, power spectra, space-time correlations, and statistics associated with the tail of distributions (extreme events). Such requirements imply high computational cost and, in general, restrict application to high-fidelity transient methods.

At each update of the physical core of the software, a systematic battery of verification tests is executed, with the objective of ensuring that fundamental properties of laminar and turbulent flows are correctly reproduced. These tests include canonical cases such as flow in a flat channel and circular duct, as well as external flow around a sphere, cylinder, and cube resting on the ground. Such configurations have analytical, semi-analytical, or consolidated experimental solutions, allowing immediate identification of numerical inconsistencies or significant physical deviations, in order to prevent structural errors from propagating to stable versions of the application.

In this section, only the validations with greater direction and relevance to wind engineering applications are presented, covering both the software and the pre- and post-processing procedures employed. Complete details of the cases mentioned here are available in our public portfolio, periodically updated, in which the conducted validation campaigns are systematically documented: portfolio.aerosim.io.
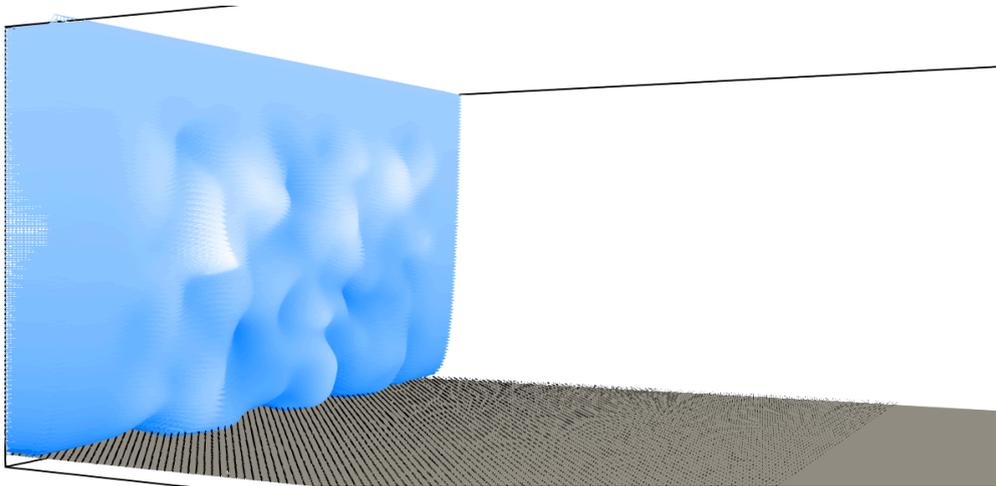
Atmospheric Boundary Layer

The atmospheric boundary layer (ABL) is the lowest portion of the atmosphere, directly influenced by friction with the surface, thermal stratification, and terrain characteristics. Within this layer, wind velocity, turbulence intensity, and length scales vary significantly with height and

surface characteristics. For wind engineering applications, the ABL is the relevant flow regime, as it governs the loads and responses experienced by structures, pedestrians, and infrastructure exposed to natural winds. Its correct replication is essential for all other wind engineering applications to be properly carried out.
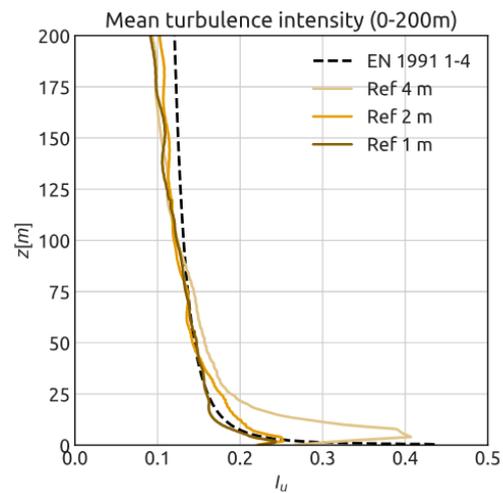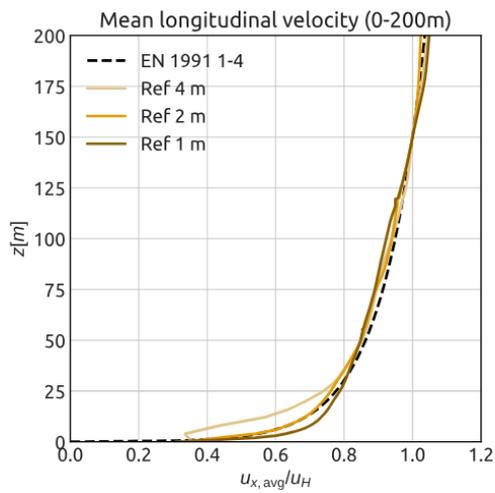

For the generation of the atmospheric boundary layer profile, a synthetic turbulence generator based on the Synthetic Eddy Method (SEM) is employed, together with roughness elements distributed along a development length. The objective is to promote the transition from the imposed inlet field to a statistically stationary state compatible with the atmospheric boundary layer, allowing adjustment of the spectral content through the energy cascade mechanism described by Kolmogorov. After this length, the useful region of the domain is established, in which buildings can be inserted and where the effective statistical properties of the flow are verified.

The following figures present the profiles of mean longitudinal velocity and turbulence intensity corresponding to the open terrain roughness category (Category II) according to EN 1991-1-4, for three mesh refinement levels. Based on these and other comparisons, systematic adjustment of spatial refinement and configuration of roughness elements is performed, in order to reproduce the reference mean profiles and turbulent statistics within previously established tolerances.

Topographic Effect

Topography introduces spatially variable modifications to the atmospheric boundary layer by inducing acceleration, deceleration, separation, and increased turbulence of the flow over hills, ridges, escarpments, and valleys. Under real conditions, these effects are inherently three-dimensional: interaction between terrain features in different directions generates complex flow patterns, including channeling, wake interference, and secondary circulations. As a result, the boundary layer structure significantly departs from flat terrain assumptions, with localized alterations in mean velocity, shear, and turbulent characteristics.


Since wind-induced pressures scale quadratically with local wind velocity, even relatively small velocity amplifications resulting from topography can produce substantial increases in pressures and structural actions. This nonlinear sensitivity is further aggravated by three-dimensional terrain interactions, in which acceleration effects associated with individual features cannot be linearly or independently superimposed.
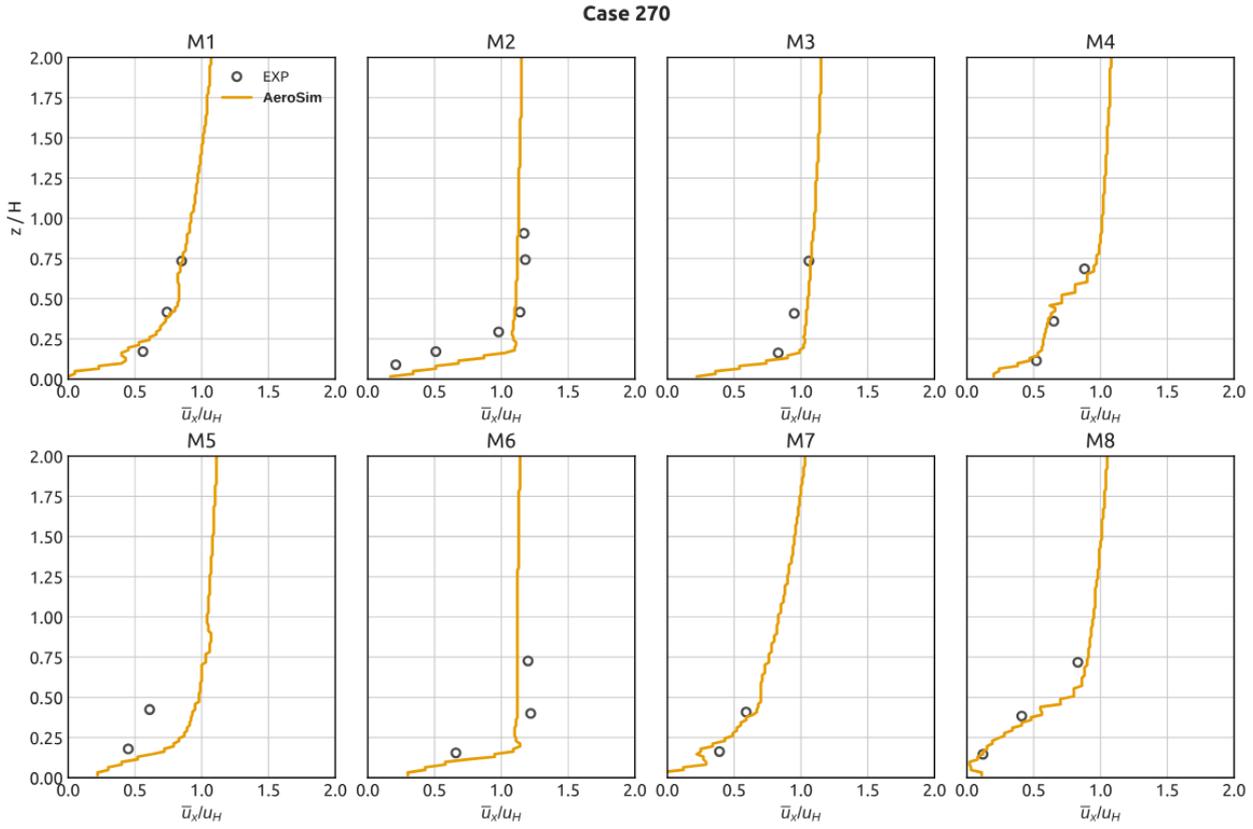
The figures below illustrate the topography of Bolund, a 12 m high island located in Roskilde Fjord, Denmark, as well as the positioning of masts with anemometers at different heights.
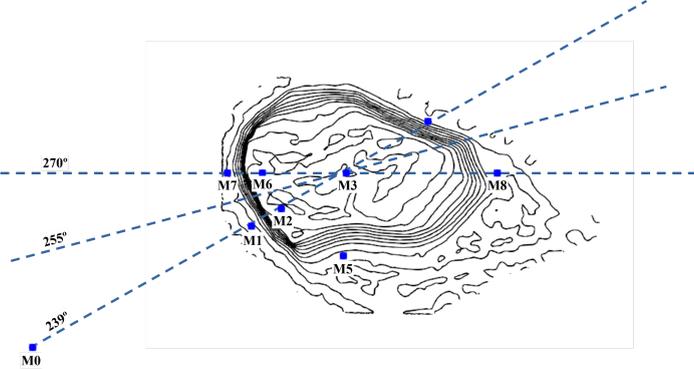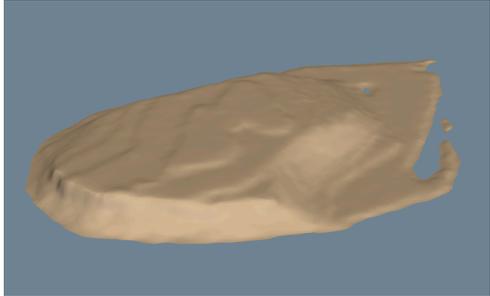
The image below presents the comparison between the topographic factor measured experimentally in the field and obtained via numerical simulation for a wind incidence of 270°.
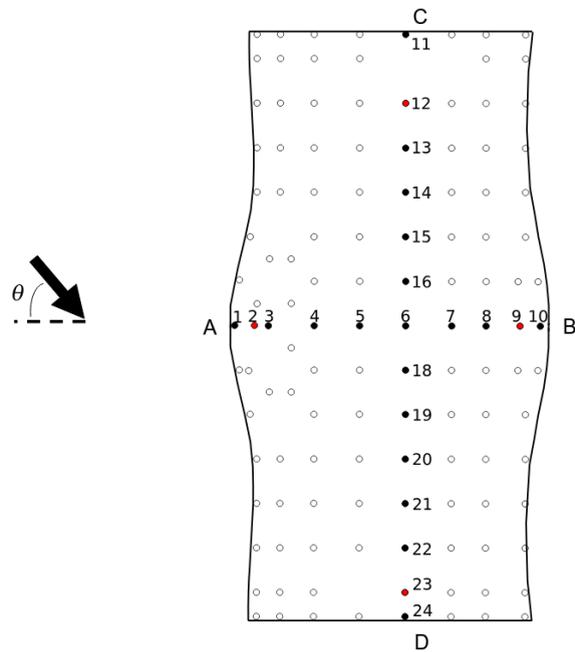
Low-Rise Buildings

Measurement of wind pressures on low-rise buildings, such as industrial facilities and warehouses, presents specific challenges. Elements such as canopies, eaves, and roof lanterns introduce sharp edges, flow separation, and reattachment zones that generate highly localized pressure fields. These effects are not adequately captured by simplified loading models and therefore require detailed measurements to resolve their spatial variability. Accurate reproduction of peak pressures is particularly critical in low-rise buildings, as structural and cladding failures are often governed by short-duration extreme suctions rather than mean loads. These peaks are strongly influenced by the temporal and spatial correlation of the flow. Thus, proper measurement must preserve not only correct mean pressures and RMS values, but also the coherence and simultaneity of pressure fluctuations between different surfaces, which directly affect resultant forces and load combinations.

The figure below presents the comparison between the experimental wind tunnel configuration and its corresponding numerical replication, considering flow over an international airport with an organic-shaped roof.
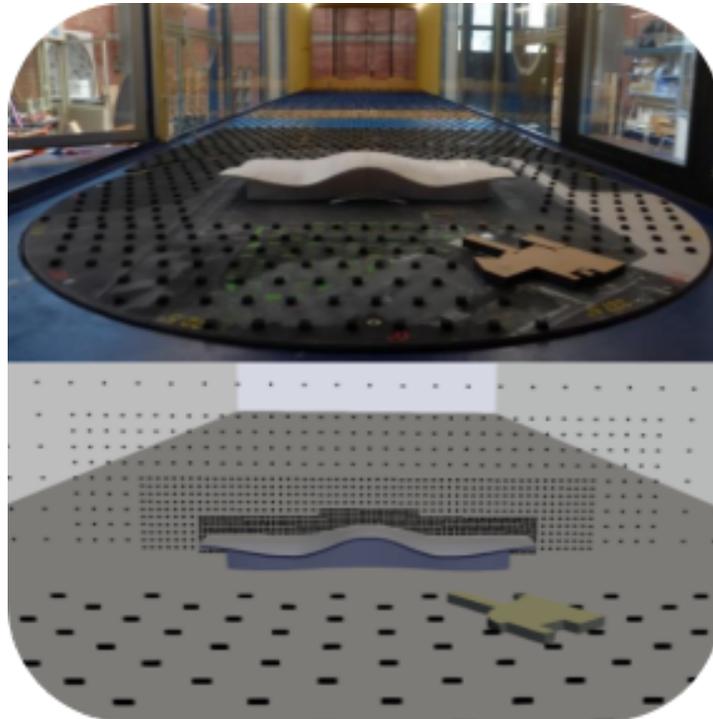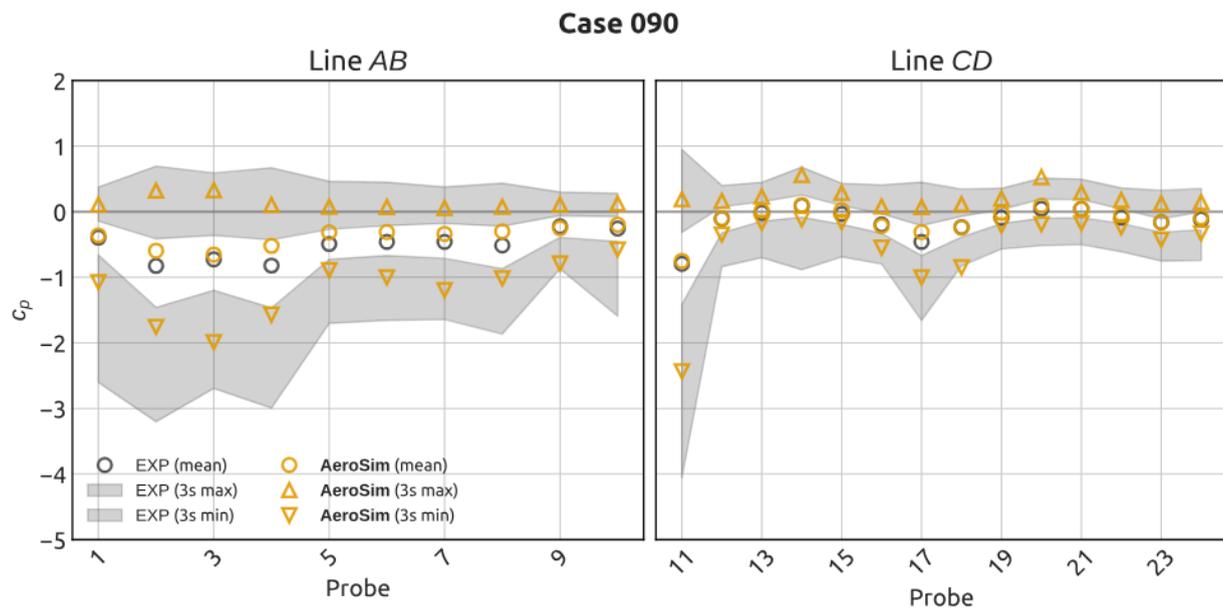
The figure below presents the distribution of pressure sensors and the system of wind incidence orientation.


The figures below present the statistics of minimum, mean, and maximum pressure coefficients for a wind incidence of 90°.

High-Rise Buildings

For high-rise buildings, wind pressure measurements constitute fundamental input for the design of façade elements, such as glazing systems, mullions, cladding panels, and anchorage components. These elements are typically governed by local peak pressures rather than global structural actions, making detailed and spatially resolved pressure data essential to ensure safety and service performance.
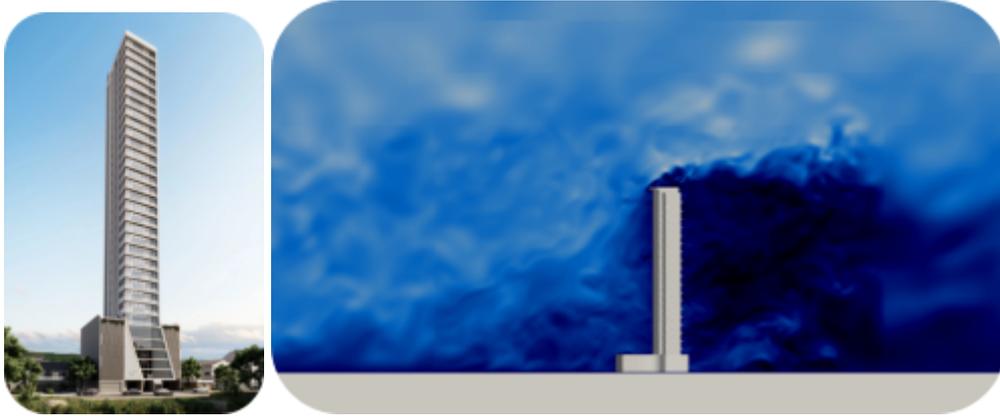
From a structural perspective, integration of wind forces per floor is a central requirement for the design of the main resisting system. Along-wind, across-wind, and torsional effects must be derived from force time histories. The dynamic response of these structures further requires reproduction of the temporal and spatial correlations of the wind field, since resonant and quasi-resonant effects directly depend on the coherence of pressure fluctuations along the height and plan of the building.

To evaluate wind effects on an isolated **complex-shaped** building, the study of a helical building conducted at the Tokyo Polytechnic University is used. The figures below present the shape and distribution of pressure sensors used.
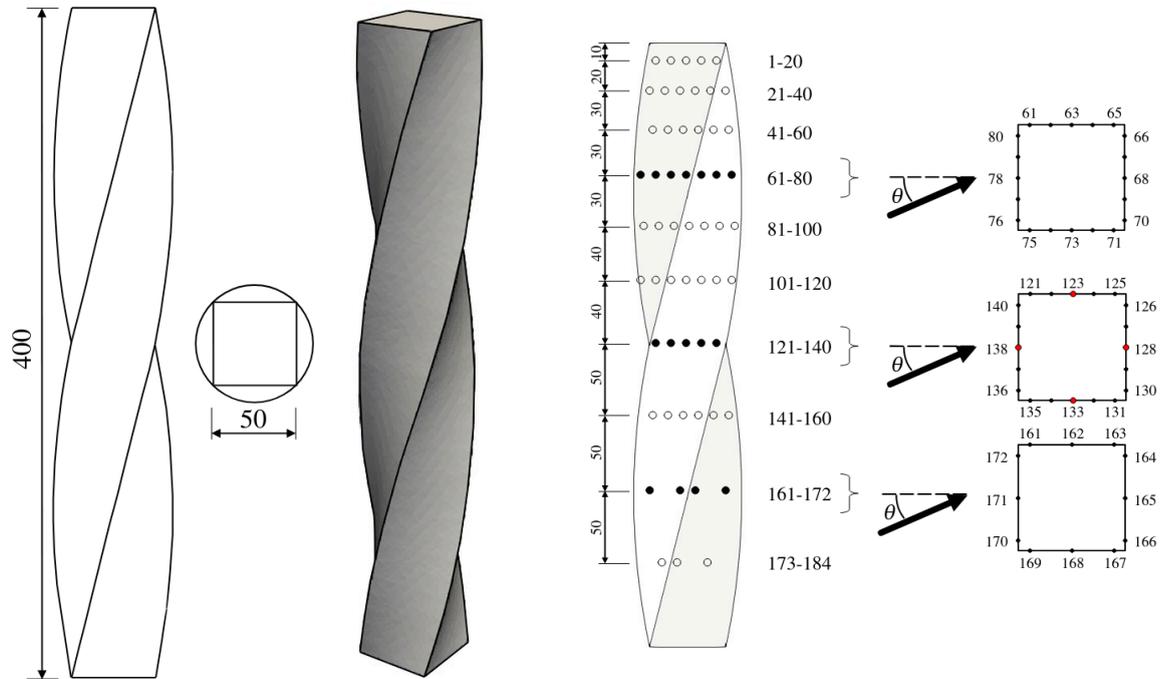
The figure below presents the comparison of minimum, mean, and maximum pressure coefficients for sensor lines positioned at 25%, 50%, and 75% of the total building height, for a wind incidence of 0°.

Alameda Cabral, 475 - São Francisco, Curitiba - PR, 80410-210

consultoria@aerosim.io

aerosim.io
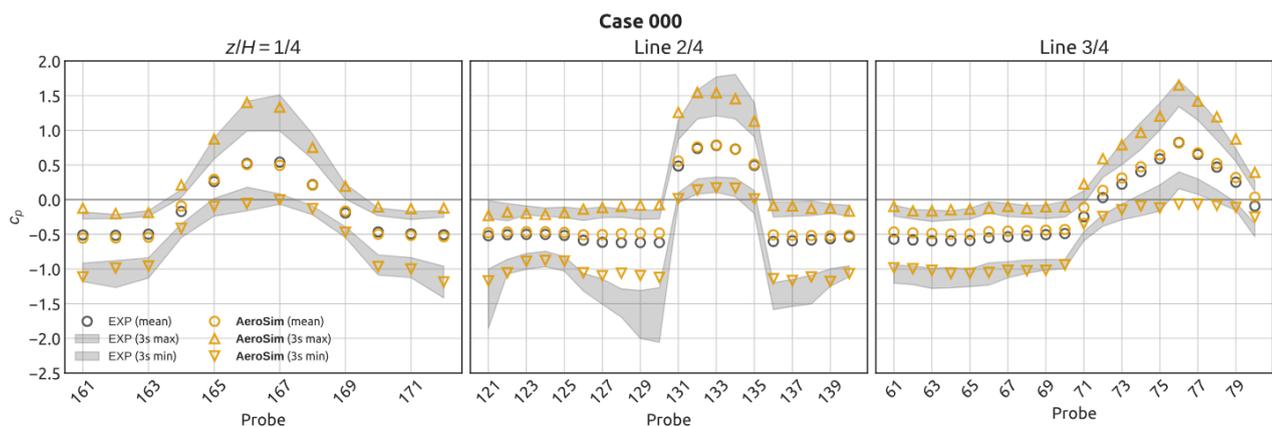
Below, the pressure coefficient spectra at selected probes are also shown, using the same wind incidence.
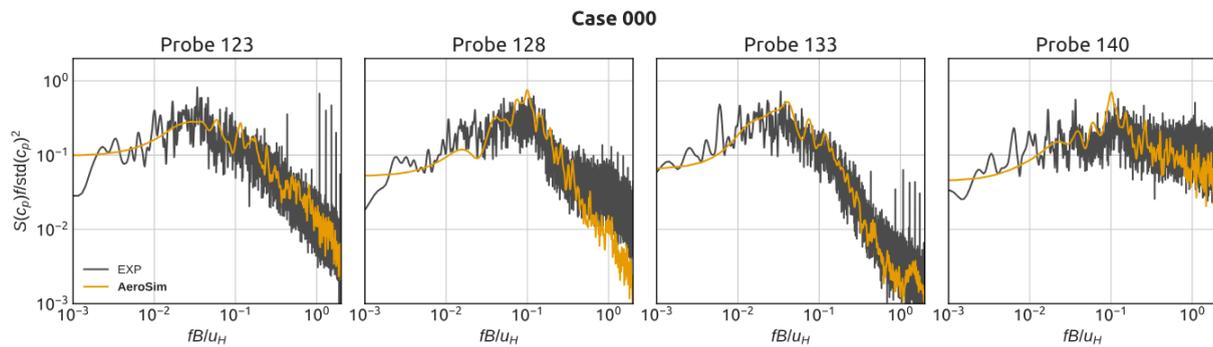
To evaluate **neighborhood effects**, another case conducted at Tokyo Polytechnic University is used, in which two flat-shaped buildings are positioned in different relative arrangements. The figures below present the neighborhood positioning and the distribution of pressure sensors.
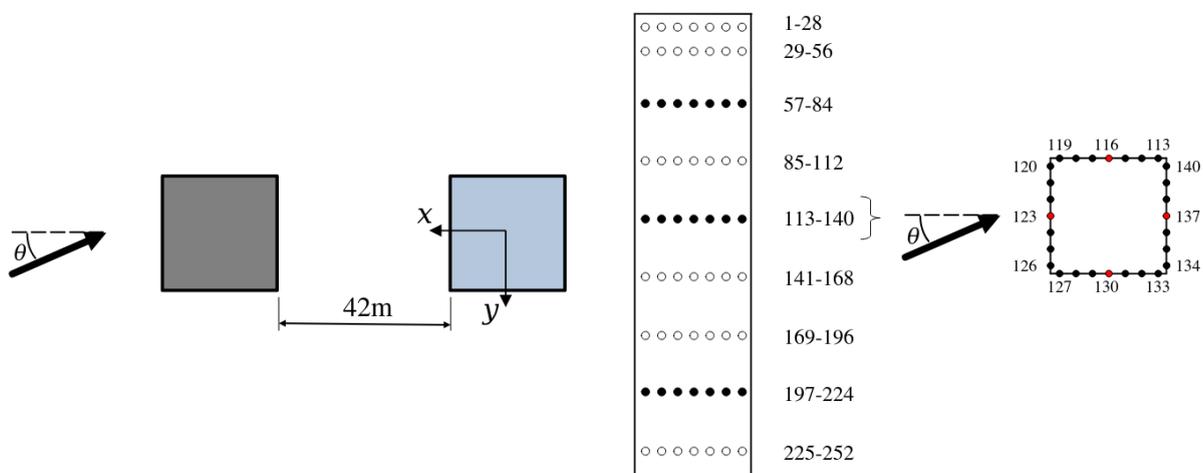
The figures below present the minimum, mean, and maximum pressures over different building heights for a 0° wind incidence, in which the main building is completely submerged in the wake of its neighbor. Pressure spectra at selected sensors are also presented.

Case 000

Case 000
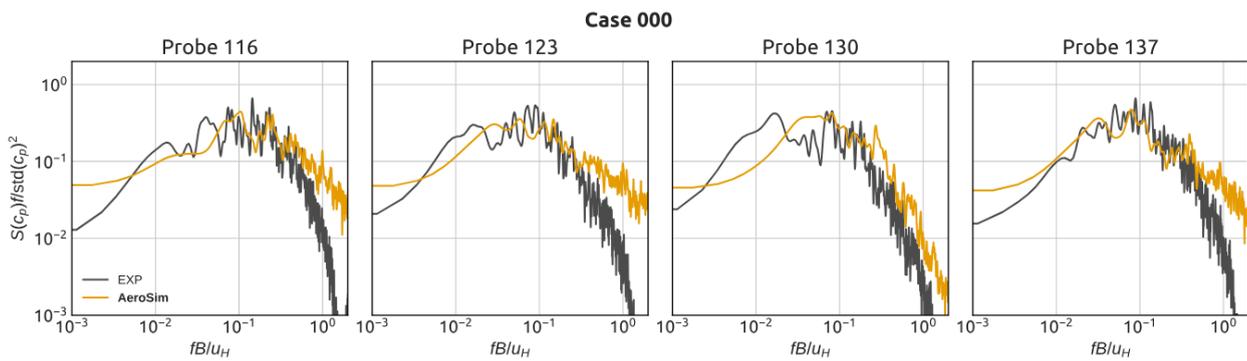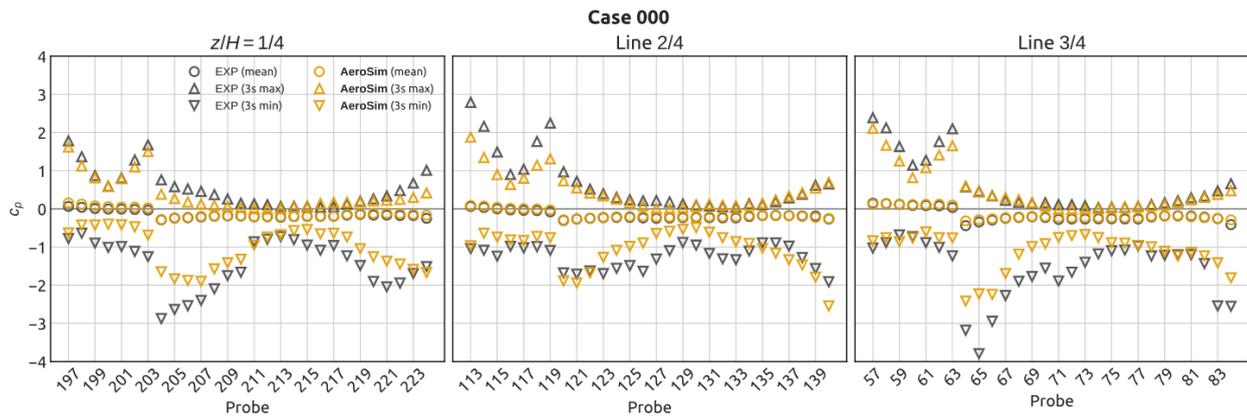
To evaluate **dynamic loads**, the result of an international comparison between multiple wind tunnels on a standardized rectangular building (CAARC building) is used. The figure below illustrates the building shape and some of the vortices generated by wind interaction.

The figures below present the comparison of bending and torsional moments at the base, for different wind incidences and 2.5% damping. The effects are converted from dynamic response to their static equivalents to be used in structural calculation.
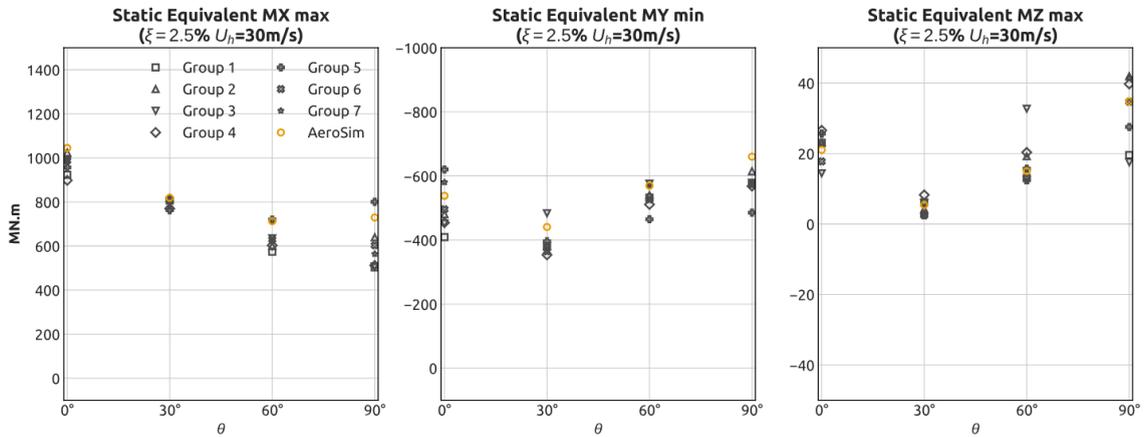
The figure below shows the acceleration obtained for the same case and 1% damping.
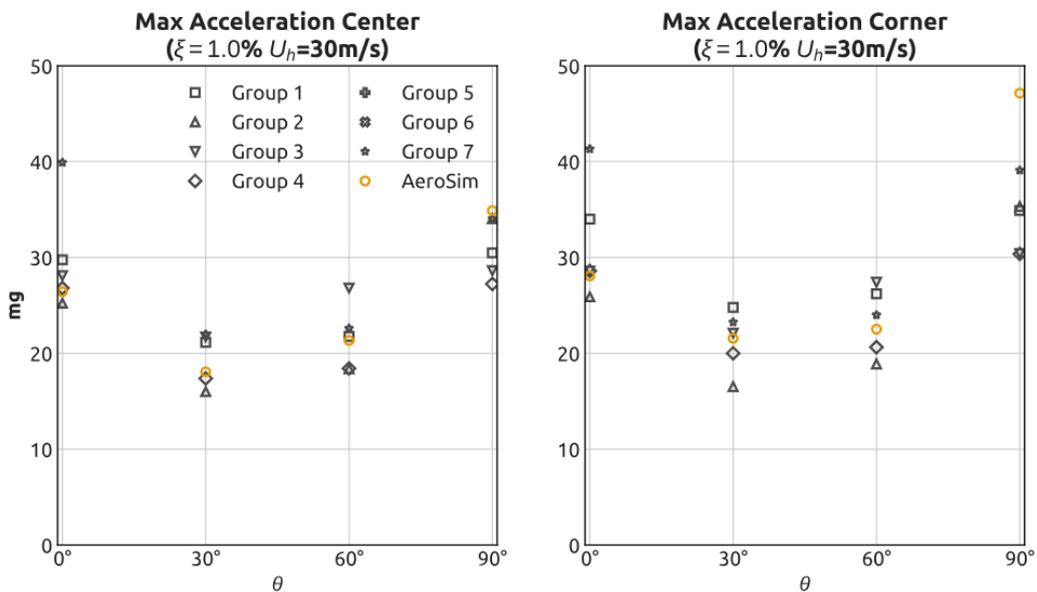
Pedestrian Comfort

Wind flow in urban environments is determined by a combination of mean velocity, turbulence intensity, gusts, directionality, and statistical occurrence of exceedance events at pedestrian height. These characteristics result from the interaction between the atmospheric boundary layer and the built environment and vary significantly at reduced spatial and temporal scales. The resulting flow conditions directly impact pedestrian comfort, as they govern both the physical forces experienced by individuals and their subjective perception of the outdoor environment.

Different urban activities, such as walking, standing, sitting, or performing outdoor leisure activities, are associated with different levels of tolerance to wind exposure, usually defined by comfort criteria that relate flow characteristics to acceptable usage conditions. High-rise buildings can significantly modify near-ground flow by deflecting high-momentum air along their façades, resulting in localized accelerations and increased turbulence. In dense urban contexts, interaction between façades, street canyons, and open spaces generates a complex three-dimensional flow field that cannot be reliably predicted by analytical methods.

To validate the application for pedestrian comfort, the case of the Architectural Institute of Japan was used, involving flow in the city of Niigata. The figure below illustrates the flow and building arrangement.
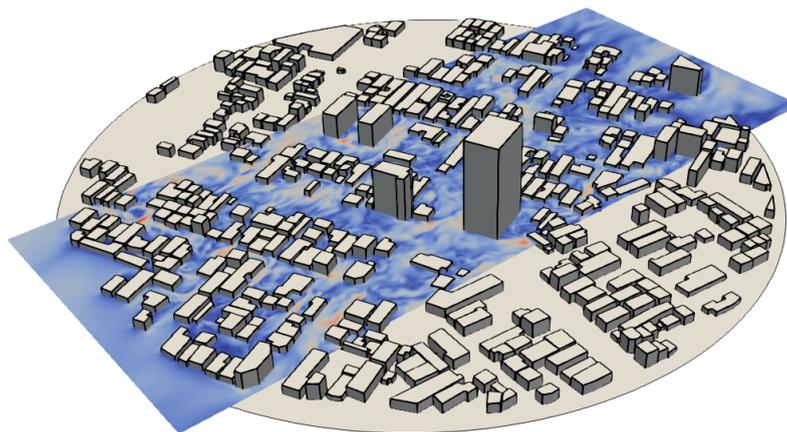
The figures below show the flow pattern resulting from north wind incidence and its comparison with experimental measurements.

Wind source - N
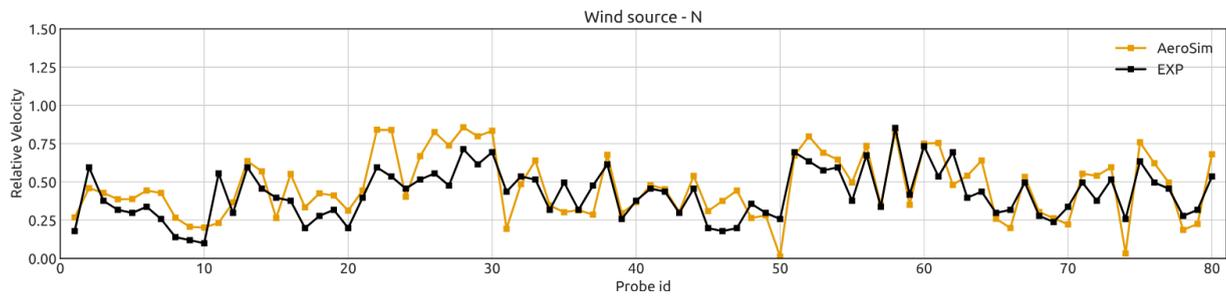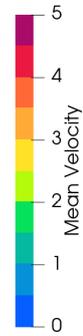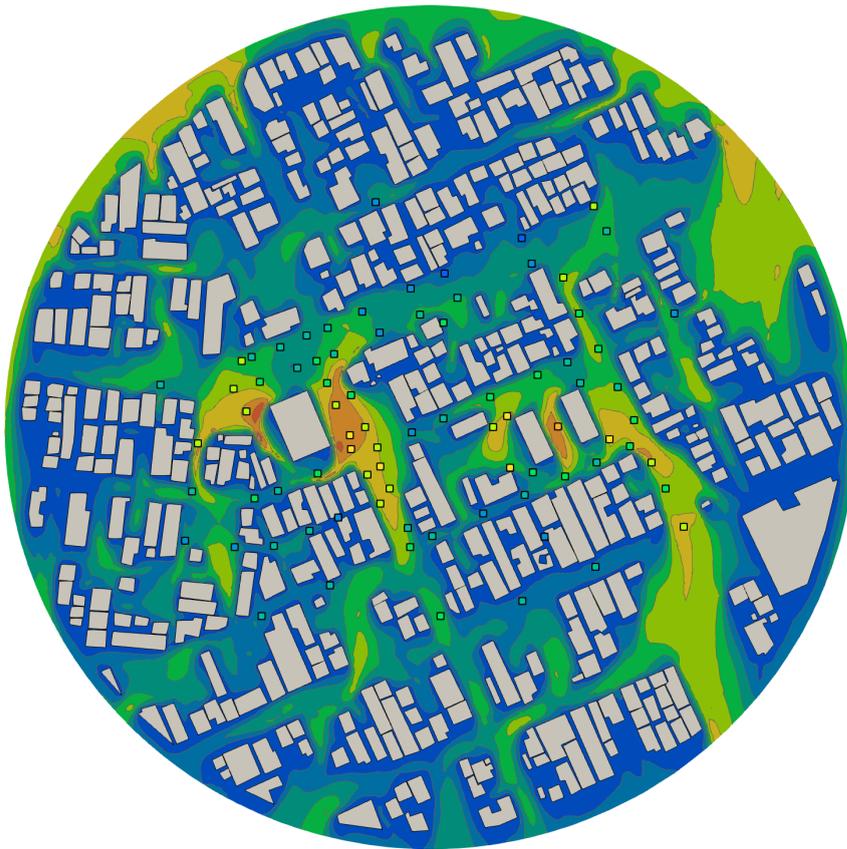
Anexo A - CFD for Wind Engineering

Experts' Position and the State of the Art

In the context of structural design - both in static and dynamic analyses - the most relevant statistics are the extreme values, defined based on the statistical properties of the flow and the admissible probability of exceedance according to design criteria. Consequently, the computational methods adopted must be capable of simulating the flow in a transient regime, with high spatial and temporal resolution, in order to adequately capture the turbulent scales up to the limit of the inertial subrange (Durbin and Reif, 2011).

A realistic representation of these turbulent structures requires the use of high-fidelity approaches such as DNS (Direct Numerical Simulation), LES (Large Eddy Simulation), DES (Detached Eddy Simulation), and SAS (Scale-Adaptive Simulation), whose computational cost is substantial. This high computational demand makes advances in computational performance a necessary condition for the feasible use of CFD in the calculation of wind loads for structural design purposes. As a result of these requirements, studies aimed at validating the application of CFD for estimating wind loads on tall buildings - through systematic comparisons between numerical and experimental results - only began to emerge more consistently after 2010 (Potsis, Tominaga and Stathopoulos, 2023), when the use of high-fidelity numerical methods combined with adequately refined meshes became viable.

A non-exhaustive list of studies conducted during this period includes: Dagnew and Bitsuamlak (2010), Huang, Li, and Li (2013), Bruno et al. (2014), Dagnew and Bitsuamlak (2014), Xie (2014), Aboshosha, Elshaer, Bitsuamlak, and El Damatty (2015), Bernardini et al. (2015), Nozu et al. (2015), Tamura et al. (2015), Zhang, Habashi, and Khurram (2015), Elshaer et al. (2016a,b), Liu and Niu (2016), Elshaer et al. (2017a,b), and Yu, Yang, and Xie (2018).

Some reviews addressing the state of the art and evaluating several of the previously cited works include: Dagnew and Bitsuamlak (2013), Hou and Jafari (2020), Daniels and Xie (2022), and Potsis, Tominaga, and Stathopoulos (2023). Based on the body of results collected to date, it is possible to draw some general conclusions, such as:

- It is definitively confirmed that the use of high-fidelity transient numerical methods is necessary for the accurate determination of wind loads applicable to structural design. Furthermore, the sensitivity of the results to spatial mesh refinement and temporal discretization has been clearly demonstrated, which imposes strict limitations on the possibility of reducing computational cost without compromising the quality of the generated data.
- The faithful reproduction of the ABL is critical, requiring not only correct profiles of mean velocity and turbulence intensity, but also their spectra and spatial correlations. For this reason, much of current research focuses on inlet boundary conditions. In many cases, it is necessary to adapt to the computational environment the techniques traditionally used in physical wind tunnels to simulate the ABL.
- Conducting simulations in CWE requires interdisciplinary technical and scientific expertise in both CFD and Wind Engineering, as well as consolidated professional experience. This is essential to ensure proper control of boundary conditions, computational modeling, and interpretation of results-analogous to what is required from a professional responsible for physical wind tunnel testing. The mere availability of CFD/CWE software or familiarity with graphical user interfaces does not qualify a professional for this role. A properly qualified professional is referred to as a CWE Specialist.
- It is not possible to establish, in a universal manner, the applicability of CFD without first identifying the class of problem to which the application belongs. The use of CFD must be preceded by the definition of the problem class-an assessment that considers multiple factors, such as the aerodynamic behavior of the structure, its typology, the specific objective of the analysis, and the design stage. Without this step, no universally valid guideline can be established.

Position of Official Standards and Guidelines Worldwide

With regard to the regulatory framework governing the use of CFD for calculating wind loads and estimating the dynamic responses of structures, there is considerable heterogeneity among different international codes. Some regulations make no explicit reference to CFD, others formally prohibit or authorize its use, while some standards adopt an informative, performance-based approach without explicitly endorsing or rejecting the technique.

The North American code **ASCE 7** – Minimum Design Loads and Associated Criteria for Buildings and Other Structures, which regulates wind load determination in Chapters 26 to 30, began to explicitly address the use of numerical simulations in its 2022 edition through additions to Chapter 31 – Wind Tunnel Procedure. In this chapter, the use of results from a "digital wind tunnel" (CFD) is permitted, provided that the adopted procedures are verifiable and validated against physical wind tunnel data and that the study is subjected to independent technical peer review.

The currently applicable code in the European Union for calculating wind actions on structures, **EN 1991-1-4:2005**, was published in 2005, with Amendment A1 added in 2010 and editorial corrections issued in 2009 and 2010. Although a new version, titled prEN 1991-1-4:2021, is under revision-with voting expected to close in July 2024-this document has not yet been published as a consolidated standard and remains in draft status.

The currently valid version establishes in Clause 1.5 – Design assisted by testing and measurements that:

> "In supplement to calculations wind tunnel tests and proven and/or properly validated numerical methods may be used to obtain load and response information, using appropriate models of the structure and of the natural wind."

and adds the note: "The National Annex may give guidance on design assisted by testing and measurements."

Among the Eurocode National Annexes, only those of Italy (CNR-DT R1-207/2018) and France (NF EN 1991-1-4/NA:2008-03) make explicit reference to the use of CFD (Computational Fluid Dynamics) for determining wind loads. The other National Annexes focus exclusively on parameters applicable to physical wind tunnel testing or analytical methods, without mentioning CFD as a permitted, prohibited, or regulated technique.

In the French case, the passage corresponding to Clause 1.5 of the National Annex (**NF EN 1991-1-4/NA:2008-03)** is relatively concise and is reproduced in full below (free translation from the original French):

> "Numerical simulation techniques for fluid flows have progressed rapidly. The governing equations (Navier–Stokes and conservation of mass) are well established, and there are methods capable of generating, at the boundaries of the computational domain, instantaneous velocity fields consistent with the statistical characteristics of natural turbulent wind (vertical gradient of mean velocity, turbulence intensity, integral scales, and spectral distribution of turbulence).
>
> The direct numerical resolution of these equations relies on an extremely fine discretization of space and time and, for this reason, remains largely beyond the reach of current computational resources when applied to civil engineering problems (such as large-scale structures).
> Statistical modeling, which lies at the core of the computational models currently used for civil engineering applications:
> - The time-averaging operation introduces new unknowns which, in order for the system to be solvable, require the introduction of complementary information in the form of physical models, of which there is a certain variety (closure relations). The quality of the simulations therefore depends on the deep expertise of the operators, both in numerical methods and in the physics of the phenomena being modeled;
> - By construction, statistical modeling seeks mean pressure values, whereas structural design depends on their extreme values. Consequently, these simulations face the limitations previously mentioned in this regard.
> Large Eddy Simulation (LES) represents a compromise between direct numerical simulation and statistical modeling. The Navier–Stokes equations are fully solved, step by step in time, for the large turbulent structures (eddies larger than the mesh size); the smaller scales are not directly simulated but are accounted for through subgrid-scale statistical models. In this way, it becomes possible to seek extreme pressure values from the generated time series. Developments in this approach are highly promising."

The Italian Annex (**CNR-DT R1-207/2018**) includes an Annex T, developed through the efforts of the ANIV-CWE working group, which presents in detail the appropriate procedures for the correct setup of simulations and the incorporation of CFD (Computational Fluid Dynamics) in the determination of wind loads within the building design process. The same content has been incorporated as Annex K in the revised version of Eurocode 1 (prEN 1991-1-4:2021). These informative annexes are primarily intended to ensure a correct, safe, and up-to-date transfer of technology between the CWE (Computational Wind Engineering) scientific community and practicing structural engineers. In essence, they were conceived to: (i) facilitate structural designers' understanding of the fundamental principles of CWE; (ii) codify good practices based on performance criteria for the use of computational methods; and (iii) warn against inappropriate uses or misinterpretations of these techniques.

This annex specifies, with a good level of detail, the fundamental principles of numerical fluid simulation and its application in wind engineering, the assumptions and simplifications adopted in geometric models and numerical schemes, the constraints regarding the dimensions of the computational model, the computational mesh, and the boundary conditions to be used.

In its Clause T.5 (Scopes of Application of CWE), the annex makes the following considerations regarding the application of computational methods to wind engineering (free translation from the Italian):

"In general, CWE is potentially applicable to a wide range of problem classes and at different design stages. On the other hand, the components of the computational model are not universal, and the overall accuracy of the numerical simulation varies significantly depending on the adopted modeling approach and numerical methods.

Therefore, the application of CWE in design is admitted with a variable role, depending on the computational approach used, the class of problem investigated, and the design stage. In general terms, the definition of this role is conditioned on the validation of the computational approach for each problem class and depends on the current state of the art in CWE, which is still undergoing rapid scientific and technical evolution. Nevertheless, this Appendix proposes in Table T.I, for indicative and synthetic purposes, a general assessment of the expected reliability level of CWE simulations.

The qualitative indicators of reliability presented are proposed based on: (i) the computational approach-simplified here by distinguishing only according to the treatment of turbulence (scale-resolving or not); (ii) the class of problem-defined solely by the expected wind effect; and (iii) the design stages-limited to the preliminary and final design phases. The estimated reliability level is also weighted by considering the potential impact that possible inaccuracies may have on the Limit State of the structure (Ultimate and/or Serviceability).

It should be noted that high reliability, extended to all problem classes, is recognized exclusively for the Scale-Resolving approach in the preliminary design phase.

It is important to emphasize that the proposed assessment assumes, in all cases, that CWE simulations are conducted in accordance with the state of the art and are not affected by human errors in the implementation and/or use of the CFD/CWE software."

Table T.I referenced indicates the reliability level of different numerical approaches for various design applications. The table presented below is equivalent to that contained in the annex, but was translated into English by Bruno et al. (2023):

Table 2 Reliability of the results obtained by CWE simulation according to the purpose of the analysis. SLS denotes Serviceability Limit States, ULS denotes Ultimate Limit States [H: High, M: Medium, L: Low, D: Discouraged, -: not envisaged].

PROBLEM CLASS / Approach to turbulence	Scale-resolving				Non scale-resolving			
Design Stage	Preliminary		Detailed		Preliminary		Detailed	
Limit State	SLS	ULS	SLS	ULS	SLS	ULS	SLS	ULS
Forces on Buildings	H	H	H	M	L	D	D	D
Forces on bridge decks	H	H	H	M	M	M	M	L
Local pressures	H	H	H	M	D	D	D	D
Vortex shedding	H	H	M	M	M	L	L	D
Galloping	H	H	M	L	M	L	L	L
Flutter	-	H	-	L	-	H	-	L
Comfort	H	-	H	-	M	-	M	-

The Japanese standard AIJ – Recommendations for Loads on Buildings (**AIJ-RLB-2015**) explicitly recognizes CFD as a valid tool for estimating wind loads. In Clause 6.1.2 (Estimation Principle), item (14), the code states (already in English in the original):

"When the computational fluid dynamics (CFD) technique is applied to the evaluation of the wind loads, it is necessary to obtain sufficient accuracy for the numerical results based on A6.11."

and further in item (16):

"When the wind load shielding effects by surrounding topographies or buildings are considered, the future changes shall be confirmed, and the shielding effect shall be investigated through the appropriate wind tunnel test, CFD, or other suitable verification methods.".

The annex A6.11 mentioned above states:

"A6.11 Wind Loads Estimation using CFD
For CFD applied to the wind load estimation, the required physical quantities are the wind velocity, velocity fluctuation, and time histories of the wind pressure or wind force. Then, the governing equations for computation should be constructed for an incompressible viscous fluid, and can express the solution as unsteady flows and wind pressures based on the engineering scale of the buildings and their surroundings. Because the wind is a turbulent flow, it cannot be obtained through an analytical solution. To obtain a numerical solution with sufficient accuracy, a discrete approximate solver and turbulent modeling should be selected appropriately."

Among the standards that explicitly prohibit the use of CFD for structural design under wind action is **ISO 4354:2009**. Although it recognizes CFD as a promising tool, the standard states that, "given the current state of development of CFD techniques, such methods are not yet capable of fully reproducing the fluctuating characteristics of the flow that are necessary to obtain the appropriate fractile of the extreme-value distribution of pressure coefficients, nor the correct correlations between fluctuating pressure coefficients over the surface, which are essential for determining force or moment coefficients over large (or global) areas." Accordingly, the standard does not recommend the use of CFD for obtaining forces and pressure coefficients. Nevertheless, it emphasizes that, should CFD be used for this purpose, its application must satisfy the same criteria required for physical wind tunnel testing, particularly regarding the faithful reproduction of the wind environment and the techniques used for analyzing results.

Possibly one of the most explicit recent codes regarding the prohibition of CFD is the Canadian National Building Code of Canada (**NBCC**), whose latest revision was issued in 2020. The code states in its Division B:

> "A-4.1.7.1.(6) Computational Fluid Dynamics (CFD). It is not currently possible to verify the reliability and accuracy of CFD and no standards address it; as such, this method is not permitted to be used to determine specified wind loads."

Within this international scenario, the Brazilian code for determining wind forces on buildings, **NBR 6123**, stands as an exception, as it makes no mention of CFD throughout its text. The only indirect parallel that could be drawn appears in the 2023 revision, Clause 1, Note 4, which states:

> "In the case of structures with shapes, dimensions, or surrounding conditions not covered by this Standard, the determination of wind forces and their effects shall be carried out with the aid of special studies involving consolidated technical, scientific, and normative knowledge and/or tests on reduced-scale physical models in a wind tunnel, with simulation of the main characteristics of natural wind.."

References

Aboshosha, H., Elshaer, A., Bitsuamlak, G. T., & El Damatty, A. (2015). **Consistent inflow turbulence generator for LES evaluation of wind-induced responses for tall buildings**. Journal of Wind Engineering and Industrial Aerodynamics, 142, 198–216.

Bernardini, E., Spence, S.M., Wei, D., Kareem, A., 2015. **Aerodynamic shape optimization of civil structures: a CFD-enabled Kriging-based approach**. J. Wind Eng. Ind. Aerod.

Braun, A. L., & Awruch, A. M. (2009b). **Aerodynamic and aeroelastic analyses on the CAARC standard tall building model using numerical simulation**. Computers & Structures, 87(9–10), 564–581.

Blocken B. **50 years of computational wind engineering: past, present and future.** Journal of Wind Engineering and Industrial Aerodynamics. 2014 Jun 1;129:69-102.

Bruno, L, Coste, N, Mannini C, Mariotti A, Patruno, L, Schito, P Vairo, G. **Codes and standards on computational wind engineering for structural design: State of art and recent trends**. Wind and Structures. 2023;37:133-51.

Bruno, L., Salvetti, M.V., Ricciardelli, F., 2014. **Benchmark on the Aerodynamics of a Rectangular 5:1 Cylinder: an overview after the first four years of activity**. J. Wind Eng. Ind. Aerod. 126, 87-106

Dagnew, A.K., Bitsuamlak, G.T., 2014. **Computational evaluation of wind loads on a standard tall building using LES**. Wind Struct. 18 (5), 567–598.

Dagnew, A. K., & Bitsuamlak, G. T. (2010). **LES evaluation of wind pressures on a standard tall building with and without a neighboring building**. 5th International Symposium on Computational Wind Engineering (CWE2010).

Dagnew, A. K., & Bitsuamlak, G. T. (2013). **Computational evaluation of wind loads on buildings: A review**. 16.

Daniels SJ, Xie ZT. **Overview of large-eddy simulation for wind loading on slender structures**. Proceedings of the Institution of Civil Engineers-Engineering and Computational Mechanics. 2022 Jun;175(2):41-71.

Durbin PA, Reif BP. **Statistical theory and modeling for turbulent flows**. John Wiley & Sons; 2011 Jun 28.

Elshaer, A., Bitsuamlak, G., Damatty, A.E., 2016a. **Aerodynamic shape optimization of tall buildings using twisting and corner modifications**. In: 8th International Colloquium on Bluff Body Aerodynamics and Applications (BBAA8). Boston, Massachusetts, USA.

Elshaer, A., Bitsuamlak, G., Damatty, A.E., 2017a. **Enhancing wind performance of tall buildings using corner aerodynamic optimization**. Eng. Struct. 136, 133–148.

Elshaer, A., Gairola, A., Adamek, K., & Bitsuamlak, G. (2017b). **Variations in wind load on tall buildings due to urban development**. Sustainable Cities and Society, 34, 264–277.

Elshaer, A., Aboshosha, H., Bitsuamlak, G.T., Damatty, A.E., Dagnew, A., 2016b. **LES evaluation of wind-induced responses for an isolated and a surrounded tall building**. Eng. Struct. 115, 179–195.

Elshaer, A., Aboshosha, H., Bitsuamlak, G., El Damatty, A., & Dagnew, A. (2016). **LES evaluation of wind-induced responses for an isolated and a surrounded tall building**. Engineering Structures, 115, 179–195.

Hou F, Jafari M. **Investigation approaches to quantify wind-induced load and response of tall buildings: A review**. Sustainable Cities and Society. 2020 Nov 1;62:102376.

Huang, S., Li, R., & Li, Q. (2013). **Numerical simulation on fluid-structure interaction of wind around super-tall building**

at high reynolds number conditions. Structural Engineering and Mechanics, 46(2), 197–212.

Liu, J., & Niu, J. (2016). **CFD simulation of the wind environment around an isolated high-rise building: An evaluation of SRANS, LES and DES models**. Building and Environment, 96, 91–106.
Nozu, T., Tamura, T., Takeshi, K., Akira, K., 2015. **Mesh-adaptive LES for wind load estimation of a high-rise building in a city**. J. Wind Eng. Ind. Aerod. 144, 62–69.

Potsis T, Tominaga Y, Stathopoulos T. **Computational wind engineering: 30 years of research progress in building structures and environment.** Journal of Wind Engineering and Industrial Aerodynamics. 2023 Mar 1;234:105346.

Tamura, T., Kawai, H., Bale, R., Onishi, K., Tsubokura, M., Kondo, K., Nozu, T., 2015. **Analysis of wind turbulence in canopy layer at large urban area using HPC database**. In: 9th International Conference on Urban Climate (ICUC9). Toulouse, France.

Thordal MS, Bennetsen JC, Koss HH. **Review for practical application of CFD for the determination of wind load on high-rise buildings.** Journal of Wind Engineering and Industrial Aerodynamics. 2019 Mar 1;186:155-68.

Xie, J., 2014. **Aerodynamic optimization of super-tall buildings and its effectiveness assessment**. J. Wind Eng. Ind. Aerod. 130, 88-98

Yu, Y., Yang, Y., & Xie, Z. (2018). **A new inflow turbulence generator for large eddy simulation evaluation of wind effects on a standard high-rise building**. Building and Environment, 138, 300–313.

Zhang, Y., Habashi, W. G., & Khurram, R. A. (2015). **Predicting wind-induced vibrations of high-rise buildings using unsteady CFD and modal analysis**. Journal of Wind Engineering and Industrial Aerodynamics, 136, 165–179.

Anexo B - Academic Publications

International Journals

Breitkopf, S. et al. Bluff Body Benchmark Consortium - Initial results of international wind tunnel and CFD study for structural design. Under Review, 2025.

Oliveira Jr, W. B. ; Romanus, R. S. ; Lugarini, A. . Nassu: A high performance LES solver for computational wind engineering. Under Review, 2025.

Ferrari, M. A. ; Oliveira Jr, W. B. ; Lugarini, A. ; Franco, A. T. ; Hegele Jr, L. A. . A graphic processing unit implementation for the moment representation of the lattice Boltzmann method. International Journal for Numerical Methods in Fluids, 95(7), 1076-1089, 2023.

Lugarini, A. ; Franco, A. T. ; Philippi, P. C. . Lattice Boltzmann method for viscoplastic fluid flow based on regularization of ghost moments. Journal of Non-Newtonian Fluid Mechanics, v. 286, p. 104413, 2020.

Romanus, R. S. ; Lugarini, A. ; Franco, A. T. . An immersed boundary-lattice Boltzmann framework for fully resolved simulations of non-spherical particle settling in unbounded domain. Computers & Mathematics With Applications, v. 102, p. 206-219, 2021.

Romanus, R. S. ; Lugarini, A. ; Franco, A. T. . Fully-resolved simulations of an ellipsoidal particle settling in a Bingham fluid. Journal of Non-Newtonian Fluid Mechanics, v. 301, p. 104745, 2022.

Congress Papers

Zavelinski, A. ; Romanus, R. S. ; Lugarini, A. ; Franco, A. T. . Estudo Comparativo de Carregamento Quasi-Estático de Vento Sobre a Cobertura de um Galpão Através de Método Transiente de Fluidodinâmica Computacional. 9o Congresso da Construção Metálica, São Paulo, 2023.

Zavelinski, A. ; Lugarini, A. ; Franco, A. T. . Utilização de fluidodinâmica computacional para calcular o fator topográfico de aceleração do vento. XIV Congresso brasileiro de pontes e estruturas, Rio de Janeiro, 2023.

Romanus, R. S. ; Lugarini, A. ; Oliveira Jr, W. B. ; Franco, A. T. . A viable framework for wind pressure assessments with large eddy simulations. 16th International conference on wind engineering, Florença, Itália, 2023.

Romanus, R. S. ; Zavelinski, A. ; Lugarini, A. . Estudo comparativo de carregamento dinâmico de vento em edifício alto através de método transiente de fluidodinâmica computacional. 64o Congresso brasileiro do concreto, 2023.

Lugarini, A. ; Romanus, R. S. ; Oliveira Jr, W. B. . Large Eddy simulations of complex shaped high-rise buildings. 9th International Colloquium on Bluff Body Aerodynamics and Applications, Birmingham, Reino Unido, 2024.

Zavelinski, A. ; Romanus, R. S. ; Lugarini, A. ; Franco, A. T. . Estudo comparativo de carregamento dinâmico de vento em edifício alto através de método transiente de fluidodinâmica computacional. 65o Congresso brasileiro do concreto, 2024.

Oliveira Jr, W. B. ; Romanus, R. S. ; Penas, P. . Normalização de Densidade em Condições de Contorno do LBM e Seu Efeito no Domínio da Frequência. IV Congresso Brasileiro de Fluidodinâmica Computacional, Vitória, 2024.

Zavelinski, A. ; Penas, P. ; Romanus, R. S. ; Lugarini, A. ; Vendramini, J. A. . Utilização de CFD para Cálculo de Pressões de Vento em Galpões sobre Taludes. XVI Congresso brasileiro de Pontes e Estruturas, Rio de Janeiro, 2025.

Oliveira Jr., W. B. ; Lugarini, A. ; Franco, A. T. . Performance analysis of the lattice Boltzmann method implementation on GPU. In: XL Ibero-Latin-American Congress on Computational Methods in Engineering, 2019, Natal, RN. Proceedings of the XL CILAMCE, 2019.

Romanus, R. S. ; Lugarini, A. ; Franco, A. T. . Numerical investigation of uniform flow past stationary sphere using immersed boundary-lattice Boltzmann method. In: XL Ibero-Latin-American Congress on Computational Methods in Engineering, 2019, Natal, RN. Proceeding of the XL CILAMCE, 2019.

Romanus, R. S. ; Lugarini, A. ; Franco, A. T. . Direct Numerical Simulation of non-Spherical Rodrigo Saldanha Romanus - rodrigo@aerosim.io

Particle Movement in an Infinite Domain Filled with Viscoplastic Fluid. In: 18th International Congress on Rheology, 2020, Rio de Janeiro, Brazil. Proceedings of 18th ICR, 2020.