

# Advances in scale-resolving simulation for turbomachinery

Oriol Lehmkuhl<sup>i</sup>, Eusebio Valero<sup>ii</sup> and Guillermo Paniagua<sup>iii</sup>

Commercial CFD solvers widely used in turbomachinery rely on steady or unsteady Reynolds-averaged Navier–Stokes (RANS/URANS) equations. These approaches, based on temporal averaging, can provide satisfactory results for simple industrial cases. However, they are severely limited when simulating the highly compressible and unsteady flows of transonic turbomachinery, where flow separation, shock–boundary layer interaction, and a broad spectrum of scales must be captured. At the other end of the spectrum, high-fidelity methods such as large-eddy simulation (LES) or direct numerical simulation (DNS) offer superior accuracy but remain computationally prohibitive for engineering design.

Recent advances in high-order discretization methods offer a promising path forward. Approaches such as discontinuous Galerkin schemes achieve higher accuracy with coarser meshes compared to second-order methods, enabling efficient resolution of multiple physical scales. When combined with turbulence-resolving strategies and robust shock-capturing techniques, still an active research frontier, these methods have the potential to deliver predictive simulations at a fraction of the cost of LES or DNS.

The advent of exascale computing further expands these possibilities. Fully exploiting architectures capable of billions of operations per second requires advances in heterogeneous hardware utilization, energy efficiency, extreme parallelism, and scalability. Numerical algorithms and software frameworks must be carefully adapted to this paradigm to balance performance and data management on a scale.

Typically, experimental data in turbomachinery is available only at discrete locations, though often with extremely fine temporal resolution—on the order of microseconds. Computational data, by contrast, can provide dense volumetric coverage of the flow field, but with greater uncertainty in representing transients. If the boundary conditions of both approaches are aligned and their uncertainties rigorously integrated, it becomes possible to construct a unified model that leverages the strengths of each, opening the possibility of new numerical accuracy.

This session will highlight progress in advanced CFD methods for turbomachinery, together with their implementation on next-generation supercomputers. Contributions are encouraged on numerical schemes, high-performance computing strategies, experimental validation, and the integration of machine learning to improve turbulence modeling and accelerate workflows. Ultimately, these efforts will enable more reliable and cost-effective design tools for industry, fostering cleaner, more efficient, and sustainable propulsion and energy systems. By bridging computational science, numerical modeling, and experimental validation, this session will provide a forum to discuss in the next generation of predictive technologies for turbomachinery.

---

<sup>i</sup> Barcelona Supercomputer Center

<sup>ii</sup> Universidad Politecnica de Madrid

<sup>iii</sup> Purdue University