24th IACM Computational Fluids Conference (CFC 2027)
March 12-15, 2027, Yokohama, Japan

COMPLEX FLUID FLOWS IN TURBOMACHINERY: SIMULATION, MODELING AND OPTIMIZATION

SHIN-ICHI TSUDA[*], SATOSHI WATANABE[**] AND JIH-HYUK KIM[†]

[*] Department of Mechanical Engineering, Kyushu University
744 Motooka, Nishi-ku, Fukuoka, Japan
tsudashin@mech.kyushu-u.ac.jp, https://hyoka.ofc.kyushu-u.ac.jp/html/100020834_en.html

[**] Department of Mechanical Engineering, Kyushu University
744 Motooka, Nishi-ku, Fukuoka, Japan
fmnabe@mech.kyushu-u.ac.jp, https://hyoka.ofc.kyushu-u.ac.jp/html/100017285_en.html

[†] Korea Institute of Industrial Technology
Cheonan 31056 Korea
jinhyuk@kitech.re.kr, https://sites.google.com/view/jinhyukkim

Key words: CFD, Complex Flow Field, Turbomachinery, Modeling, Optimization

ABSTRACT

In turbomachinery—such as compressors, turbines, pumps, and hydroturbines—turbulent flow fields characterized by highly complex vortex behaviours are formed. Furthermore, at an operating point far from the design condition, flow fields with strong unsteadiness or instability often develop due to phenomena such as rotating stall or surge. When flows involving condensation or evaporation occur in turbomachinery—such as droplet formation in steam turbines or cavitation in pumps and hydroturbines—the resulting flow phenomena become even more complex, and the performance and unsteadiness of the turbomachinery change significantly. As application range of turbomachinery expands, the thermodynamic conditions under which working fluids are used are becoming increasingly diverse. A recent characteristic feature is the growing difficulty in predicting phenomena caused by physical properties that differ significantly from those of water or air, such as supercritical flow of carbon dioxide and the use of cryogenic fluids like liquid hydrogen. Therefore, while it is an extremely important work to accurately predict the various flow phenomena that can occur in turbomachinery using advanced CFD and to link these findings to modelling and optimizations that can be fed back into the design process, the reality is that we are still only halfway there.

The objective of this Minisymposium is to share the latest numerical analysis and computational techniques across various fields of turbomachinery, enabling participants to broadly envision future research directions without being confined to specific computational methods or subject areas. We plan to accept presentations on studies aimed at gaining a detailed understanding of flow fields through advanced CFD analysis or that with fluid-structure interaction, modelling of internal flow phenomena, or optimization calculations, focusing primarily on rotating machinery—including representative air and hydraulic machinery, as well as marine propellers. We also actively welcome topics utilizing digital twin technology based on experimental findings or employing AI. Furthermore, studies on flow fields in fundamental geometries—such as those around an airfoil/hydrofoil or within simple flow channels—that are closely

related to the flows in turbomachinery, are also welcome. Not only researchers specializing in turbomachinery but also researchers or young birds/students from other fields can apply for this Minisymposium.