Simulation of pressure loads with parallelized Direct Surface Description solver

Qwist, Jesper Roland Kjærgaard; Christensen, Erik Damgaard

Publication date:
2023

Document Version
Publisher's PDF, also known as Version of record

Danish Offshore Technology Centre
Technology Conference 2023

Simulation of pressure loads with parallelized Direct Surface Description solver

Jesper Roland Kjærgaard Qwist and Erik Damgaard Christensen

The Direct Surface Description method is implemented in the open-source computational fluid dynamics code OpenFOAM® (www.openfoam.com). The first version implemented in OpenFOAM-v1912 only allowed single core simulations, which meant that real world cases of adequate resolution were not computationally feasible. Therefore, our latest efforts have been dedicated to the parallelization of the first implementation of Direct Surface Description method. This has resulted in a significant refactoring of the code to accommodate the needed parallel communication. Additionally, the code has been updated to work with OpenFOAM-v2306, which is the newest release from June 2023. The poster will present simulated pressure loads on structures using the new parallelized Direct Surface Description solver.

AARHUS UNIVERSITY
UNIVERSITY OF COPENHAGEN
DTU
GEUS
AALBORG UNIVERSITY
DENMARK