



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

[Link back to DTU Orbit](#)

Citation (APA):
Qwist, J. R. K., & Christensen, E. D. (2023). *Simulation of pressure loads with parallelized Direct Surface Description solver*. Abstract from Danish Offshore Technology Conference 2023 , Kolding, Denmark.

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal

If you believe that this document breaches copyright please contact us providing details, and we will remove access to the work immediately and investigate your claim.

Danish Offshore Technology Centre
Technology Conference 2023

Simulation of pressure loads with parallelized Direct Surface Description solver

Jesper Roland Kjærgaard Qvist 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

