solids4foam-v2.1: A toolbox for performing solid mechanics and fluid-solid interaction simulations in OpenFOAM

Philip Cardiff [1,¶], **Ivan Batisti'c** [2], **and eljko Tukovi'c** [2]

1 School of Mechanical and Materials Engineering, University College Dublin, Dublin, Ireland **2** Faculty of Mechanical Engineering and Naval Architecture, University of Zagreb, Zagreb, Croatia ¶ Corresponding author

Summary

`solids4foam` is a toolbox for performing solid mechanics and fluid-solid interaction simulations within the popular OpenFOAM (ESI-OpenCFD, 2024; foam-extend, 2024; Foundation, 2024) software. The `solids4foam` toolbox offered a rich set of features, including fluid-solid and thermo-fluid-solid interaction coupling algorithms, a suite of solid material models, advanced solid boundary conditions (e.g. frictional contact, fracture), and several discretisations (e.g., cell-centred, vertex-centred) and solution algorithms (coupled, segregated, explicit) for solid mechanics.

Statement of need

The `solids4foam` toolbox has been designed to address four primary needs: - There is a desire to solve fluid-solid interactions in OpenFOAM; - There is a desire to solve advanced solid mechanics problems natively in OpenFOAM; - There is a need for a modular approach for coupling different solid and fluid procedures in OpenFOAM; - There is a desire for an extendible framework for research into novel finite volume methods for solid mechanics.

In addition, four principles have been followed in the design of the `solids4foam` toolbox: - If you can use OpenFOAM, you can use solids4foam; - The three main OpenFOAM forks are supported by `solids4foam`: OpenFOAM.com, OpenFOAM.org, and foam-extend; - Easy toolbox should be easy to install and minimise additional dependencies beyond the requirements of OpenFOAM; - A significant emphasis is placed on code design and code style, following the OpenFOAM coding style guide closely.

Acknowledgements

References

ESI-OpenCFD. (2024). *OpenFOAM.com*. ESI Group. https://www.openfoam.com

foam-extend. (2024). *Foam-extend project*. https://sourceforge.net/projects/foam-extend/

Foundation, T. O. (2024). *OpenFOAM.org*. https://www.openfoam.org