

OpenCMP: An Open-Source Computational Multiphysics Package

Elizabeth Julia Monte¹, Alexandru Andrei Vasile¹, James Lowman¹, and Nasser Mohieddin Abukhdeir^{12¶}

¹ Department of Chemical Engineering, University of Waterloo, Ontario, Canada ² Department of Physics and Astronomy, University of Waterloo, Ontario, Canada ¶ Corresponding author

DOI: [10.xxxxxx/draft](https://doi.org/10.xxxxxx/draft)

Software

- [Review](#) ↗
- [Repository](#) ↗
- [Archive](#) ↗

Editor: [Juanjo Bazán](#) ↗

Submitted: 30 November -001

Published: unpublished

License

Authors of papers retain copyright and release the work under a Creative Commons Attribution 4.0 International License ([CC BY 4.0](#)).

Summary

OpenCMP is a computational multiphysics software package based on the finite element method (Ferziger & Perić, 2002). It is primarily intended for physicochemical processes in which fluid convection plays a significant role. OpenCMP uses the NGSolve finite element library (Schöberl, 2020) for spatial discretization and provides a configuration file-based interface for pre-implemented models and time discretization schemes. It also integrates with Netgen (Schöberl, 2020) and Gmsh (Geuzaine & Remacle, 2009) for geometry construction and meshing. Additionally, it provides users with built-in functionality for post-processing, error analysis, and data export for visualisation using Netgen (Schöberl, 2020) or ParaView (Ahrens et al., 2005).

OpenCMP development follows the principles of ease of use, performance, and extensibility. The configuration file-based user interface is intended to be concise, readable, and intuitive. Furthermore, the code base is structured and documented (Monte, Elizabeth J, 2021) such that experienced users with appropriate background can add their own models with minimal modifications to existing code. The finite element method enables the use of high-order polynomial interpolants for increased simulation accuracy, however, continuous finite element methods suffer from stability and accuracy (conservation) for fluid convection-dominated problems. OpenCMP addresses this by providing discontinuous Galerkin method (Cockburn et al., 2000) solvers, which are locally conservative and improve simulation stability for convection-dominated problems. Finally, OpenCMP implements the diffuse interface or diffuse domain method [Nguyen et al. (2018)], which a type of continuous immersed boundary method (Mittal & Iaccarino, 2005), which allows complex simulation domains to be meshed by non-conforming structured meshes for improved simulation stability and reduced computational complexity (under certain conditions (Monte et al., 2021)).

Statement of Need

Simulation-based analysis continues to revolutionize the engineering design process. Simulations offer a fast, inexpensive, and safe alternative to physical prototyping for preliminary design screening and optimization. Simulations can also offer more detailed insight into the physical phenomena of interest than is feasible from experimentation. Computational multiphysics is an area of particular importance since coupled physical and chemical processes are ubiquitous in science and engineering.

However, there are several barriers to more wide spread use of simulations. One such barrier is the lack of user-friendly finite element-based open-source simulation software. Many of the most widely-used computational multiphysics software packages, such as COMSOL Multiphysics (COMSOL Multiphysics®, n.d.) or ANSYS Fluent (Ansys® Fluent, n.d.), are closed-source

and cost-prohibitive. Furthermore, the predominance of the finite volume method for fluid dynamics simulations inherently limits simulation accuracy for a given mesh compared to the finite element method (Ferziger & Perić, 2002). Many high-quality open-source packages exist which use the finite element method, such as the computational multiphysics software MOOSE (Permann et al., 2020) or the finite element libraries NGSolve (Schöberl, 2020) and FEniCS (Alnaes et al., 2015). However, they require that the user has a relatively detailed understanding of the finite element method, such as derivation of the weak formulation of the problem, along with programming background. Alternatively, open-source finite volume-based packages such as OpenFOAM and SU2 are more accessible to the broader scientific and engineering community in that they require a less detailed understanding of numerical methods and programming, instead requiring an understanding of the command line interface (CLI) and configuration through a set of configuration files.

A second barrier is the complex geometries inherent to many real-world problems. Conformal meshing of these complex geometries is time and labour intensive and frequently requires user-interaction, making conformal mesh-based simulations infeasible for high-throughput design screening of many industrially-relevant processes (Yu et al., 2015). A possible solution is the use of immersed boundary methods (Mittal & Iaccarino, 2005) to allow the use of non-conforming structured meshes - simple and fast to generate - for any geometry. Use of structured meshes can also potentially improve simulation stability compared to unstructured meshes (Yu et al., 2015). The diffuse interface has been shown by Monte et al (Monte et al., 2021) to significantly speed-up inherently low accuracy simulations - such as those used for preliminary design screening and optimization - compared to conformal mesh-based simulations. Providing an easy-to-use publicly available implementation of this method would enable broader use by the research community and further development.

The goal of OpenCMP is to fill this evident need for an open-source computational multiphysics package which is user-friendly, based on the finite element method, and which implements the diffuse interface method. OpenCMP is built on top of the NGSolve finite element library (Schöberl, 2020) to take advantage of its extensive finite element spaces, high performance solvers, and preconditioners. OpenCMP provides pre-implemented models and a configuration file-based user interface in order to be accessible to the general simulation community, not just finite element experts. The user interface is designed to be intuitive, readable, and requires no programming experience - solely knowledge of the CLI. Users must choose the model that suits their application, but need no experience with the actual numerical implementation of said model. Finally, OpenCMP provides a publicly available implementation of the diffuse interface method with support for stabilized Dirichlet boundary conditions (Nguyen et al., 2018).

Features

The table below summarizes the current capabilities of OpenCMP. The numerical solvers for each of these models have been verified using common benchmarks (Monte, Elizabeth J, 2021). Future work on OpenCMP will focus on adding models - for multi-phase flow, turbulence, and heat transfer - and enabling running simulations in parallel over multiple nodes with MPI (Forum, 2015).

Feature	Description
Meshing	Accepts Netgen (Schöberl, 2020) or Gmsh (Geuzaine & Remacle, 2009) meshes
Numerical Methods	Standard continuous Galerkin finite element method Discontinuous Galerkin finite element method Diffuse interface method
Models	Poisson (Heat) equation

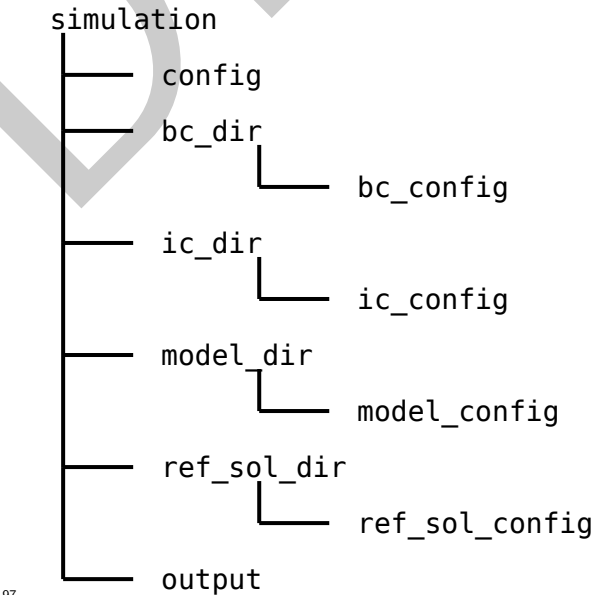
Feature	Description
Time Schemes	Stokes equations
	Incompressible Navier-Stokes (INS) equations
	Multicomponent Mixture INS equations
	First-, second-, and third- order discretizations
Solvers	Adaptive time-stepping
	Direct or iterative solvers
	Direct, Jacobi, or multigrid preconditioners
Post-Processing	Oseen (Cockburn et al., 2003) or IMEX (Ascher et al., 1995)
	linearization of nonlinear models
	Error norms calculated based on reference solutions
	Mesh and polynomial refinement convergence tests
	General simulation parameters (surface traction, divergence of velocity...)
Performance	Exports results to Netgen (Schöberl, 2020) or ParaView (Ahrens et al., 2005) format
	Multi-threading

Further information, including [installation instructions](#) and [tutorials](#) can be found on the OpenCmp [website](#). The tutorials are intended to guide new users through the various features offered in OpenCmp. Notes on the [mathematical foundations](#) of the various models and [code documentation](#) are also provided.

Software testing is provided through integration tests, which confirm the accuracy of the implemented models and time discretization schemes, and unit tests which currently offer 72% line coverage. Further information regarding performance verification of OpenCmp may be found in ref. ([Monte, Elizabeth J, 2021](#)).

User Interface

Drawing inspiration from packages like [OpenFOAM](#) and [SU2](#), the OpenCmp user interface is organized around configuration files and the CLI. Each simulation requires its own directory to hold its configuration files and outputs. This is known as the run directory or run_dir/. The standard layout of this directory is shown below.



98 The main directory and each subdirectory contain a configuration file `< >_config`. These are
99 plaintext files that specify the simulation parameters and run conditions.

100 The configuration file in the main directory contains general information about the simulation
101 including which model, mesh, finite element spaces, and solver should be used. It also contains
102 information about how the simulation should be executed such as the level of detail in the
103 output messages and the number of threads to use.

104 The `bc_dir/` subdirectory contains information about the boundary conditions. Its configuration
105 file specifies the type and value of each boundary condition. This subdirectory also contains
106 files describing boundary condition data if a boundary condition value is to be loaded from file
107 instead of given in closed form.

108 The `ic_dir` subdirectory holds information about the initial conditions. Its configuration file
109 specifies the value of the initial condition for each model variable. Like `bc_dir/`, `ic_dir/` may
110 contain additional files from which the initial condition data is loaded during the simulation.

111 The `model_dir/` subdirectory contains information about model parameters and model func-
112 tions. Its configuration file specifies the values of any model parameters or functions for each
113 model variable and the subdirectory may hold additional data files to be loaded during the
114 simulation.

115 The `ref_sol_dir/` subdirectory contains information about the error analysis to be conducted
116 on the final simulation result. Its configuration file specifies what error metrics should be
117 computed during post-processing. This configuration file also contains the reference solutions
118 the results should be compared against, either in closed form or as references to other files in
119 the subdirectory that are loaded during post-processing.

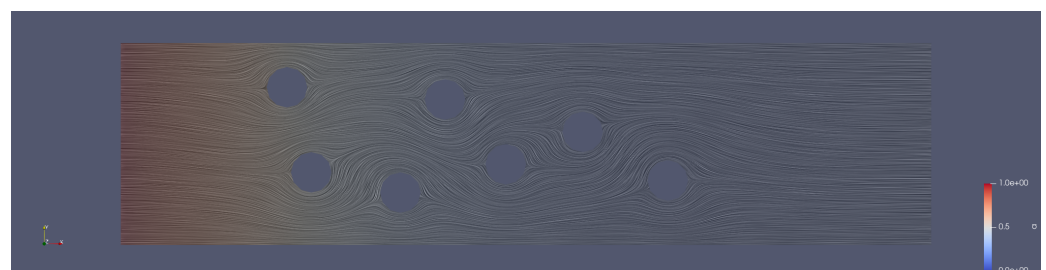
120 The `output/` subdirectory contains the saved simulation data. It does not need to be created
121 before running the simulation, it will be generated automatically if results should be saved to
122 file.

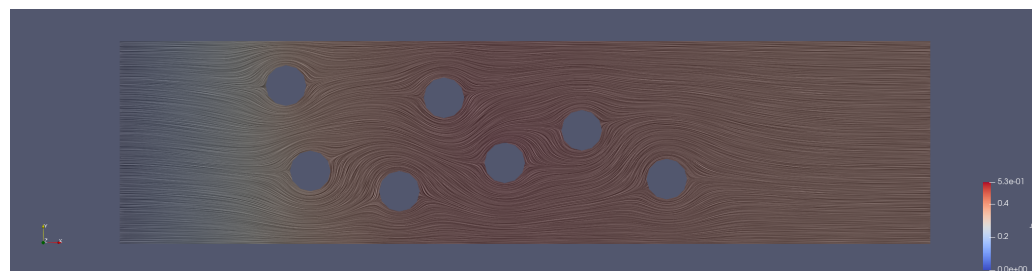
123 Examples of Usage

124 Several examples of usage of OpenCMP are present in ref. (Monte, Elizabeth J, 2021) and also
125 available via the its [website](#). Three relevant examples are summarized here: multi-component
126 incompressible flow (transient 2D, [Tutorial 8](#)), using the diffuse interface method to approximate
127 complex geometries (steady-state 3D, [Tutorial 9](#)), and incompressible flow around an immersed
128 cylinder (steady-state 3D, [Tutorial 10](#)).

129 [Tutorial 8](#) demonstrates the usage of OpenCMP to solve a multiphysics problem, two-
130 dimensional transient multi-component incompressible flow around a set of immersed circular
131 objects within a rectangular channel, shown below.

132 The mixture is composed of two components (A, B) where A undergoes an irreversible reaction
133 $A \rightarrow B$. A parabolic inlet flow of pure A is imposed, such that the mixture undergoes convection,
134 reaction, and molecular diffusion throughout the channel. Sample simulation results of the
135 steady-state are shown resulting from a cosine ramp of the inlet velocity an initial condition
136 with no flow and pure A.

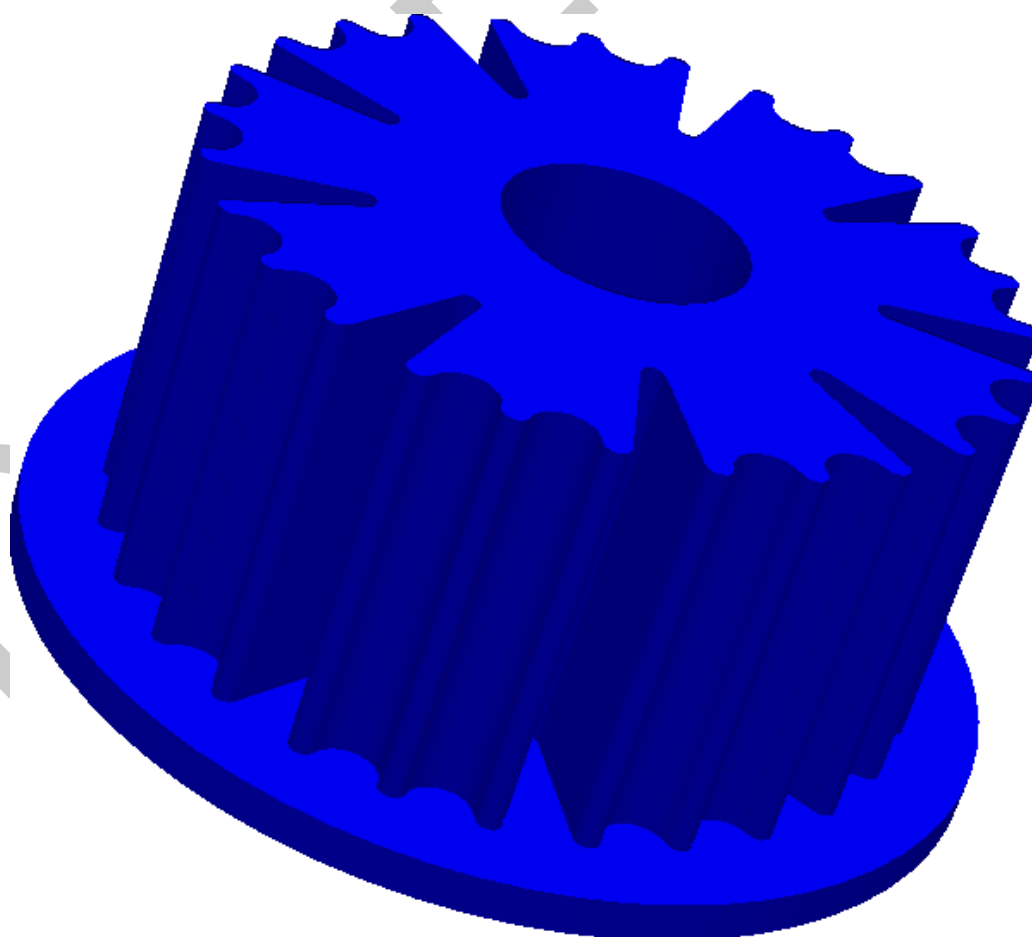




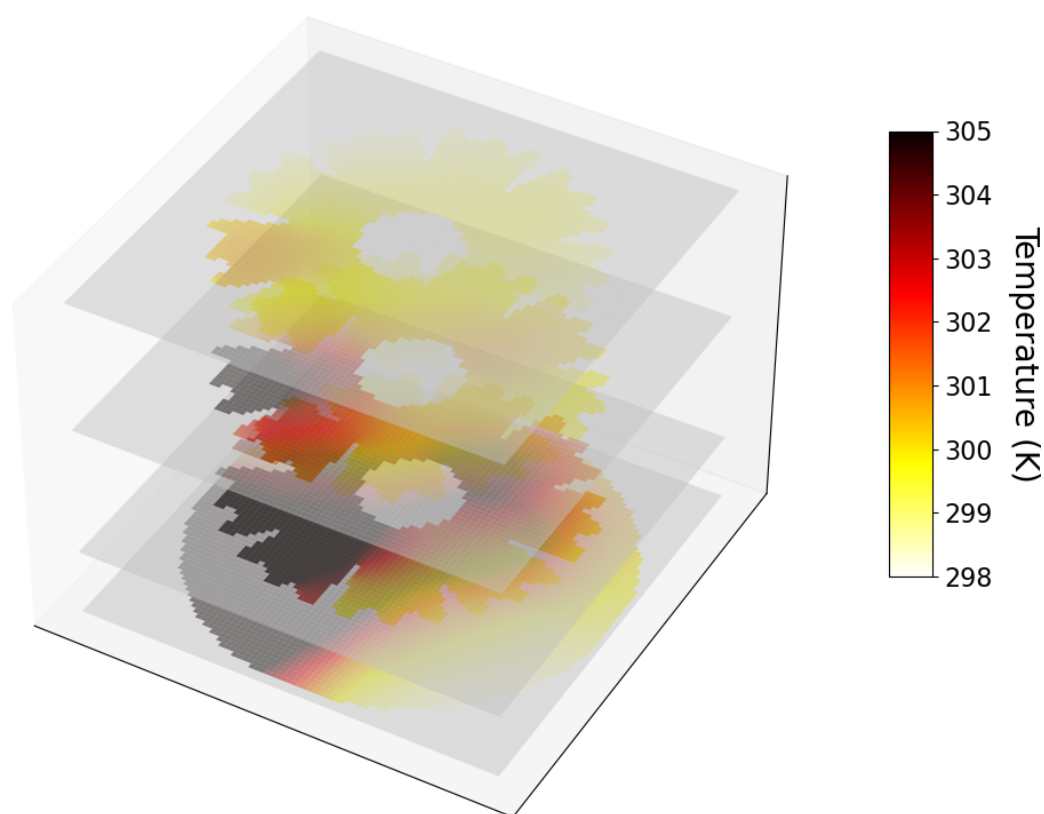
138

139 **Tutorial 9** demonstrates how to use the diffuse interface method to approximate complex
140 geometries with nonconformal structured quadrilateral/hexahedral meshes, versus the use of a
141 traditional unstructured mesh which conforms to the complex geometry boundary. The sample
142 problem is based on simulation of heat transfer within an LED heat sink from ref. (Monte et
143 al., 2021), with a geometry shown below.

144 The base of the heat sink is exposed to a spatially varying heat flux profile, corresponding
145 to heat generated by an LED assembly, with convective heat transfer conditions assumed on
146 the exterior fins. A script is provided for ease of post-processing visualization, showing the
147 diffuse-interface boundaries (comparable to the geometry above) and steady-state temperature
148 profile.

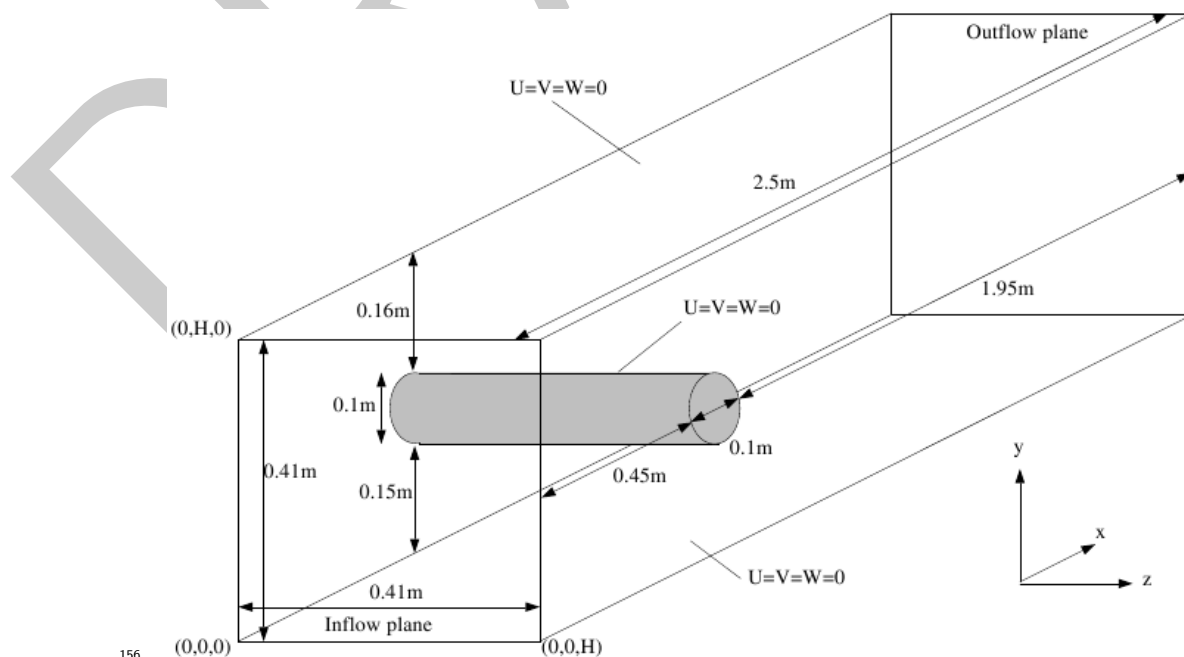


149



150

Tutorial 10 demonstrate using OpenCMP to perform a standard benchmark for 3D flow around an immersed cylinder (Bayraktar et al., 2012) under laminar flow conditions. Both continuous and discontinuous Galerkin finite element solver configurations are provided, defaulting to the continuous Galerkin variation due to reduced memory constraints. The geometry is shown below with parabolic inlet velocity resulting in $Re=20$.



156

This example also uses the built-in error analysis functionality of OpenCMP to automatically compute the force vector on the immersed cylinder through integration of the surface traction

over its boundary. This facilitates the calculation of the resulting drag and lift forces on the immersed cylinder. A visualization of the three-dimensional steady-state velocity field is shown below.



Several additional examples of usage of OpenCMP in tutorial form are available via the [website](#).

Acknowledgements

The authors would like to thank Prof. Sander Rhebergen for useful discussions regarding the discontinuous Galerkin method and Prof. Joachim Schöberl for useful discussions regarding IMEX time integration, preconditioning, and usage of the NGSolve finite element library. This research was supported by the Natural Sciences and Engineering Research Council (NSERC) of Canada and Compute Canada.

References

- Ahrens, J., Geveci, B., & Law, C. (2005). *ParaView: An end-user tool for large data visualization* (Technical Report LA-UR-03-1560). Los Alamos National Laboratory. <https://doi.org/10.1016/b978-012387582-2/50038-1>
- Alnaes, M., Blechta, J., Hake, J., Johansson, A., Kehlet, B., Logg, A., Richardson, C., Ring, J., Rognes, M. E., & Wells, G. N. (2015). The FEniCS Project Version 1.5. *Archive of Numerical Software*, 3. <https://doi.org/10.11588/ans.2015.100.20553>
- Ansys[®] Fluent. (n.d.). ANSYS Inc. Online. <https://www.ansys.com/products/fluids/ansys-fluent>
- Ascher, U. M., Ruuth, S. J., & Wetton, B. T. R. (1995). Implicit-explicit methods for time-dependent partial differential equations. *SIAM Journal on Numerical Analysis*, 32(3), 797–823. <https://doi.org/10.1137/0732037>
- Bayraktar, E., Mierka, O., & Turek, S. (2012). Benchmark computations of 3D laminar flow around a cylinder with CFX, OpenFOAM and FeatFlow. *International Journal of Computational Science and Engineering*, 7(3), 253–266. <https://doi.org/10.1504/ijcse.2012.048245>
- Cockburn, B., Kanschat, G., & Schötzau, D. (2003). The local discontinuous Galerkin method for the Oseen equations. *Mathematics of Computation*, 73(246), 569–593. <https://doi.org/10.1090/s0025-5718-03-01552-7>
- Cockburn, B., Karniadakis, G. E., & Shu, C.-W. (2000). *Discontinuous Galerkin methods: Theory, computation and applications* (1st ed., pp. 3–50). Springer-Verlag. ISBN: 3-540-66787-3
- COMSOL Multiphysics[®]. (n.d.). COMSOL AB; Online. <https://www.comsol.com/>
- Ferziger, J. H., & Perić, M. (2002). *Computational methods for fluid dynamics* (3rd ed.). Springer-Verlag. ISBN: 3-540-42074-6

- 195 Forum, M. P. I. (2015). *MPI: A message-passing interface standard version 3.1*. High-
196 Performance Computing Center.
- 197 Geuzaine, C., & Remacle, J.-F. (2009). Gmsh: A three-dimensional finite element mesh gener-
198 ator with built-in pre- and post-processing facilities. *International Journal for Numerical*
199 *Methods in Engineering*, 79(11), 1309–1331. <https://doi.org/10.1002/nme.2579>
- 200 Mittal, R., & Iaccarino, G. (2005). Immersed boundary methods. *Annual Review of Fluid*
201 *Mechanics*, 37, 239–261. <https://doi.org/10.1146/annurev.fluid.37.061903.175743>
- 202 Monte, E. J., Lowman, J., & Abukhdeir, N. M. (2021). A diffuse interface method for
203 simulation-based screening of heat transfer processes with complex geometries. *The*
204 *Canadian Journal of Chemical Engineering*. <https://doi.org/10.1002/cjce.24320>
- 205 Monte, Elizabeth J. (2021). *OpenCMP: An open-source computational multiphysics package*
206 [Master's thesis, UWSpace]. <http://hdl.handle.net/10012/17239>
- 207 Nguyen, L. H., Stoter, S. K. F., Ruess, M., Sanchez Uribe, M. A., & Schillinger, D. (2018).
208 The diffuse Nitsche method: Dirichlet constraints on phase-field boundaries. *Int. J. Numer.*
209 *Methods Eng.*, 113(4), 601–633. <https://doi.org/10.1002/nme.5628>
- 210 Permann, C. J., Gaston, D. R., Andrš, D., Carlsen, R. W., Kong, F., Lindsay, A. D., Miller,
211 J. M., Peterson, J. W., Slaughter, A. E., Stogner, R. H., & Martineau, R. C. (2020).
212 MOOSE: Enabling massively parallel multiphysics simulation. *SoftwareX*, 11, 100430.
213 <https://doi.org/10.1016/j.softx.2020.100430>
- 214 Schöberl, J. (2020). *Netgen/NGSolve*. Online. <https://ngsolve.org/>
- 215 Yu, W., Zhang, K., & Li, X. (2015, July). Recent algorithms on automatic hexahedral
216 mesh generation. *The 10th International Conference on Computer Science & Education*.
217 <https://doi.org/10.1109/iccse.2015.7250335>