

BioDEG

Biodegradation and Corrosion Simulation using Finite Element

User Manual

Version 0.8

(generated August 31, 2021)

Mojtaba Barzegari
Liesbet Geris

[BioDeg website](#)

KU LEUVEN

Copyright (c) 2019-2021 University of Leuven and [BioDeg authors](#).

Contents

1	Introduction	3
1.1	Authors	3
1.2	Acknowledgments	3
1.3	Referencing BIODEG	3
2	Useful background information	3
3	Installation	4
3.1	Compiling and installing external libraries	4
3.1.1	Instructions for PETSc and Qt	4
3.1.2	Instructions for FreeFEM	4
3.2	Obtaining and Compiling BIODEG	4
4	Running BioDeg	4
4.1	Configuring the simulation	4
4.1.1	Example 1	4
4.1.2	Example 2	4
5	Future extensions to BioDeg	4
6	Finding answers to more questions	4
A	Run-time input parameters	4
A.1	Geometry and mesh parameters	5
A.2	Materials and boundary conditions parameters	6
A.3	Solver parameters	8
A.4	Output parameters	10
	Index of run-time parameters with section names	12

1 Introduction

BIODEG is an open source software written in FreeFEM (a domain-specific language for finite element programming), C++, and Python for modeling the degradation of metallic biomaterials and simulating the biodegradation behavior of medical devices, implants and scaffolds in corrosion experiments. It can handle any geometry of desire and supports parallel computing to simulate models with a high level of complexity.

1.1 Authors

BIODEG is developed by the [Biomechanics Research group at KU Leuven and University of Liege](#). The code is currently maintained by its principal developer, who manage the development of the mathematical models and the core functionalities.

Principal developer

- Mojtaba Barzegari (University of Leuven, Belgium).

Previous Contributors

- Yann Guyot (University of Liege, Belgium).
- Piotr Bajger (University of Oxford, UK).

Mentor

- Liesbet Geris (University of Leuven, Belgium).

1.2 Acknowledgments

The development of BIODEG open source code is financially supported by the Prosperos project, funded by the Interreg VA Flanders – The Netherlands program, CCI grant no. 2014TC16RFCB046 and by the Fund for Scientific Research Flanders (FWO), grant G085018N. The developers also acknowledge support from the European Research Council under the European Union's Horizon 2020 research and innovation programmes, ERC CoG 772418.

1.3 Referencing BioDeg

Please refer to [BIODEG repository](#), section "Publications and referencing" to properly cite the use of BIODEG in your scientific work.

2 Useful background information

You may refer to the following articles for a background of the methods and algorithms implemented in BIODEG.

In addition, below are some useful references on finite element method and some online resources that provide a background of finite elements and their application to the solution of partial differential equations.

3 Installation

All the underlying installation instructions assume a Linux operating system. We assume standard tools and libraries like CMake, compilers- (C, C++ and Fortran), and MPI libraries are pre-installed. Most high-performance computers would have the latest version of these libraries in the default environment.

3.1 Compiling and installing external libraries

3.1.1 Instructions for PETSc and Qt

3.1.2 Instructions for FreeFEM

3.2 Obtaining and Compiling BioDeg

Assuming that you have already installed the above external dependencies, next follow the steps below to obtain and compile BIODeg.

4 Running BioDeg

4.1 Configuring the simulation

4.1.1 Example 1

4.1.2 Example 2

5 Future extensions to BioDeg

The future versions of BioDeg will focus on the implementation of the following methodologies/features.

-

6 Finding answers to more questions

If you have questions that go beyond this manual, there are a number of resources:

- For questions/suggestions about BIODeg installation, bugs, or similar stuff please use the [BIODEG issue tracker](#).
- BIODeg is primarily based on the [FreeFEM](#). If you have particular questions about FreeFEM, contact the community at <https://community.freefem.org/>.
- If you have specific questions about BIODeg that are not suitable for public and archived mailing lists, you can contact the primary developer and mentor:
 - Mojtaba Barzegari: mojtaba.barzegari@kuleuven.be.
 - Liesbet Geris: liesbet.geris@kuleuven.be (Mentor).

A Run-time input parameters

The underlying description of the input parameters also includes a “Standard/Advanced” label, which signifies whether an input parameter is a standard one or an advanced level parameter. The default values of the “Advanced” parameters are good enough for almost all cases. However, in some cases user may need to use “Advanced” labeled parameters. For user convenience, all input parameters are also indexed at the end of this manual in Section [A.4](#).

A.1 Geometry and mesh parameters

- *Parameter name:* `mesh_file`
Default: Should be provided
Description: [Standard]
Possible values: Any string value
- *Parameter name:* `label_scaffold`
Default: 1
Description: [Standard]
Possible values: Any positive integer value
- *Parameter name:* `label_medium`
Default: 2
Description: [Standard]
Possible values: Any positive integer value
- *Parameter name:* `label_wall`
Default: 3
Description: [Advanced]
Possible values: Any positive integer value
- *Parameter name:* `label_inlet`
Default: 4
Description: [Advanced]
Possible values: Any positive integer value
- *Parameter name:* `label_outlet`
Default: 5
Description: [Advanced]
Possible values: Any positive integer value
- *Parameter name:* `box_length`
Default: 20.0
Description: [Standard]
Possible values: Any positive floating point number
- *Parameter name:* `cube_size_x`
Default: 13.0
Description: [Standard]
Possible values: Any positive floating point number
- *Parameter name:* `cube_size_y`
Default: 13.0
Description: [Standard]
Possible values: Any positive floating point number

- *Parameter name:* `cube_size_z`
Default: 4.0
Description: [Standard]
Possible values: Any positive floating point number
- *Parameter name:* `mesh_size`
Default: 32
Description: [Standard]
Possible values: Any positive integer number
- *Parameter name:* `refine_initial_mesh`
Default: false (0)
Description: [Advanced]
Possible values: A boolean value (1 or 0)
- *Parameter name:* `mshmet_error`
Default: 0.01
Description: [Advanced]
Possible values: Any floating point number
- *Parameter name:* `mesh_size_min`
Default: 0.04
Description: [Advanced]
Possible values: Any floating point number
- *Parameter name:* `mesh_size_max`
Default: 0.8
Description: [Advanced]
Possible values: Any floating point number

A.2 Materials and boundary conditions parameters

- *Parameter name:* `material_density`
Default: 1.735e-3
Description: [Standard]
Possible values: Any floating point number
- *Parameter name:* `film_density`
Default: 2.3446e-3
Description: [Standard]
Possible values: Any floating point number
- *Parameter name:* `material_satur`
Default: 0.134e-3
Description: [Advanced]
Possible values: Any floating point number

- *Parameter name:* `material_eps`
Default: 0.55
Description: [Advanced]
Possible values: Any floating point number between 0 and 1
- *Parameter name:* `material_tau`
Default: 1.0
Description: [Advanced]
Possible values: Any floating point number
- *Parameter name:* `d_mg`
Default: 0.05
Description: [Standard]
Possible values: Any floating point number
- *Parameter name:* `d_cl`
Default: 0.05
Description: [Standard]
Possible values: Any floating point number
- *Parameter name:* `d_oh`
Default: 25.2
Description: [Standard]
Possible values: Any floating point number
- *Parameter name:* `k1`
Default: 7.0
Description: [Standard]
Possible values: Any floating point number
- *Parameter name:* `k2`
Default: 1e15
Description: [Standard]
Possible values: Any floating point number
- *Parameter name:* `fluid_nu`
Default: 0.85
Description: [Advanced]
Possible values: Any floating point number
- *Parameter name:* `fluid_in_x`
Default: 0.1
Description: [Advanced]
Possible values: Any floating point number

- *Parameter name:* `fluid_in_y`
Default: 0
Description: [Advanced]
Possible values: Any floating point number
- *Parameter name:* `fluid_in_z`
Default: 0
Description: [Advanced]
Possible values: Any floating point number
- *Parameter name:* `initial_cl`
Default: 5.175e-6
Description: [Standard]
Possible values: Any floating point number
- *Parameter name:* `initial_oh`
Default: 1e-7
Description: [Standard]
Possible values: Any floating point number

A.3 Solver parameters

- *Parameter name:* `solve_mg`
Default: true (1)
Description: [Standard]
Possible values: Any boolean value (1 or 0)
- *Parameter name:* `solve_film`
Default: true (1)
Description: [Standard]
Possible values: Any boolean value (1 or 0)
- *Parameter name:* `solve_cl`
Default: true (1)
Description: [Standard]
Possible values: Any boolean value (1 or 0)
- *Parameter name:* `solve_oh`
Default: true (1)
Description: [Standard]
Possible values: Any boolean value (1 or 0)
- *Parameter name:* `solve_ls`
Default: true (1)
Description: [Standard]
Possible values: Any boolean value (1 or 0)

- *Parameter name:* `solve_fluid`
Default: false (0)
Description: [Standard]
Possible values: Any boolean value (1 or 0)
- *Parameter name:* `solve_full_ns`
Default: true (1)
Description: [Advanced]
Possible values: Any boolean value (1 or 0)
- *Parameter name:* `write_fluid_output`
Default: true (1)
Description: [Advanced]
Possible values: Any boolean value (1 or 0)
- *Parameter name:* `solve_fluid_each`
Default: 10
Description: [Advanced]
Possible values: Any positive integer number
- *Parameter name:* `time_step`
Default: 0.025
Description: [Advanced]
Possible values: Any floating point number
- *Parameter name:* `final_time`
Default: 21.0
Description: [Standard]
Possible values: Any floating point number
- *Parameter name:* `do_redistance`
Default: true (1)
Description: [Advanced]
Possible values: Any boolean value (1 or 0)
- *Parameter name:* `redistance_time`
Default: 1.0
Description: [Advanced]
Possible values: Any floating point number

A.4 Output parameters

- *Parameter name:* `text_output_file`
Default: "output/result.txt"
Description: [Standard]
Possible values: Any string value referring to a valid path
- *Parameter name:* `write_vtk`
Default: true (1)
Description: [Standard]
Possible values: Any boolean value (1 or 0)
- *Parameter name:* `vtk_output_name`
Default: "output/output"
Description: [Standard]
Possible values: Any string value referring to a valid path
- *Parameter name:* `save_each`
Default: 0.25
Description: [Standard]
Possible values: Any floating point number
- *Parameter name:* `save_last_state`
Default: true (1)
Description: [Standard]
Possible values: Any boolean value (1 or 0)
- *Parameter name:* `output_per_area`
Default: false (0)
Description: [Advanced]
Possible values: Any boolean value (1 or 0)
- *Parameter name:* `save_multiplier`
Default: 1.0
Description: [Advanced]
Possible values: Any floating point number
- *Parameter name:* `export_scaffold`
Default: false (0)
Description: [Advanced]
Possible values: Any boolean value (1 or 0)
- *Parameter name:* `export_scaffold_each`
Default: 1.0
Description: [Advanced]
Possible values: Any floating point number

- *Parameter name:* `export_scaffold_volume`
Default: true (1)
Description: [Advanced]
Possible values: Any boolean value (1 or 0)
- *Parameter name:* `export_scaffold_surface`
Default: true (1)
Description: [Advanced]
Possible values: Any boolean value (1 or 0)
- *Parameter name:* `save_initial_mesh`
Default: false (0)
Description: [Advanced]
Possible values: Any boolean value (1 or 0)
- *Parameter name:* `save_initial_partitioned_mesh`
Default: false (0)
Description: [Advanced]
Possible values: Any boolean value (1 or 0)

Index of run-time parameters with section names

The following is a listing of all run-time parameters, sorted by the section in which they appear.

Geometry and mesh

- box length, [5](#)
- cube size x, [5](#)
- cube size y, [5](#)
- cube size z, [6](#)
- label inlet, [5](#)
- label medium, [5](#)
- label outlet, [5](#)
- label scaffold, [5](#)
- label wall, [5](#)
- mesh file, [5](#)
- mesh size, [6](#)
- mesh size max, [6](#)
- mesh size min, [6](#)
- mshmet error, [6](#)
- refine initial mesh, [6](#)

Materials and boundary conditions

- d cl, [7](#)
- d mg, [7](#)
- d oh, [7](#)
- film density, [6](#)
- fluid in x, [7](#)
- fluid in y, [8](#)
- fluid in z, [8](#)
- fluid nu, [7](#)
- initial cl, [8](#)
- initial oh, [8](#)
- k1, [7](#)
- k2, [7](#)
- material density, [6](#)
- material eps, [7](#)

- material satur, [6](#)

- material tau, [7](#)

Output

- export scaffold, [10](#)
- export scaffold each, [10](#)
- export scaffold surface, [11](#)
- export scaffold volume, [11](#)
- output per area, [10](#)
- save each, [10](#)
- save initial mesh, [11](#)
- save initial partitioned mesh, [11](#)
- save last state, [10](#)
- save multiplier, [10](#)
- text output file, [10](#)
- vtk output name, [10](#)
- write vtk, [10](#)

Solver

- do redistance, [9](#)
- final time, [9](#)
- redistance time, [9](#)
- solve cl, [8](#)
- solve film, [8](#)
- solve fluid, [9](#)
- solve fluid each, [9](#)
- solve full ns, [9](#)
- solve ls, [8](#)
- solve mg, [8](#)
- solve oh, [8](#)
- time step, [9](#)
- write fluid output, [9](#)