

Contents

1	Prerequisites	1
1.1	MUMPS	1
1.2	Configuration and compilation	1
1.3	Generation of the mesh and post-processing of the results	1
2	Walkthrough	2
2.1	Creation of the geometry	2
2.2	Creation of the mesh	2
2.3	Distribution of the electrical conductivity	3
2.4	Configuration and solving	3

Chapter 1

Prerequisites

1.1 MUMPS

The discretization of the induction equation by means of a Galerkin finite element method generates a sparse system of equations where the unknowns are the real and imaginary parts of the induced electric field. This system of equations is solved by the parallel direct solver MUMPS, used here in its sequential version. Depending on your configuration, MUMPS might require the additional libraries Metis and MPI.

These settings are tuned by the variables *MUMPS_PATH*, *METIS_PATH* and *MPI_PATH* in the configuration file *config.cmake*.

1.2 Configuration and compilation

The configuration of the compilation environment is done through CMake. The CMake configuration file *config.cmake* must be adapted to your needs and possibilities. Create a new directory *build* and, in a terminal having access to your compiler, go to that new directory. The configuration and generation of the makefiles is done through the command

```
cmake.exe -G Ninja -DCMAKE_MAKE_PROGRAM="path_to_ninja/ninja.exe" ..
```

Here the make program **ninja** has been used. Other possibilities exist (Visual Studio, etc.) and the user is free to choose. The programs are compiled and linked with the command

```
cmake.exe --build .
```

1.3 Generation of the mesh and post-processing of the results

The mesh generator and result post-processor exploited by this project is **Gmsh**. The present solver reads the mesh from the *.msh* files as well as the distribution of the electrical conductivity generated by the program *EXE/build_initial_solution*.

Chapter 2

Walkthrough

This chapter explains the different steps from the mesh generation to the launch of the solver. The steps are

- Creation of the geometry in a *.geo* file (Gmsh).
- Creation of the mesh in a *.msh* file (Gmsh).
- Use the program *EXE/build_initial_solution* to generate the distribution of the electrical conductivity inside the torch (and occasionally in its downstream region).
- Configure the parameters of the computation.
- Launch the computation of the electric fields.
- Exploit the results with the help of Gmsh.

2.1 Creation of the geometry

The geometry of the ICP torches is very simple because they are mostly cylinders which can be represented as rectangles in an axisymmetric system of coordinates. The geometry of the external domain can be freely chosen (rectangle, circle, etc.). Example of such a geometry is given in the file *Grids/Mesh_wo_coils_rectangle.geo*. The data can be modified in the file itself and reloaded in the Gmsh software. It should be stressed that the different regions of the computational domain are identified through Physical Tags (see the end of the *.geo* file). The external domain and the interior of the torch have different Physical Tags. The segments that form the wall of the torch and the axis have different tags. These tags are used in the programs in order to rapidly select the region of interest (for example to apply a particular boundary condition on a given edge).

2.2 Creation of the mesh

The configuration of the mesh is provided in an example file *Grids/Mesh_wo_coils_rectangle.geo*. Just execute the mesh building with the Gmsh software and export the resulting mesh in a *.msh* file (format 4.1).

2.3 Distribution of the electrical conductivity

The program *EXE/build_initial_solution* reads the mesh generated in the previous step and builds the distribution of the electrical conductivity, used as an input by the solver. The program is launched by the command

```
EXE/build_initial_solution mesh_input.msh mesh_output_with_sigma.msh
```

Feel free to modify the file *SRC/Build_initial_condition.f90* to your needs. Please note the usage of the physical tag explained in the previous section in order to select the interior of the torch (in the subroutine *write_initial_condition_gmsh*).

2.4 Configuration and solving

All the parameters required by the solver *EXE/icp* are put in a file *parameters*, which contains the following lines

```
1 mesh_with_sigma.msh          : name of mesh file that contains sigma, gmsh format
2 solution_output.msh          : name of the output, gmsh format
3 3                             : number of boundary types, the next 3 lines
4 1                             : Far field, physical tag in gmsh
5 2                             : Axis, physical tag in gmsh
6 3                             : Torch, physical tag in gmsh
7 150.0                         : Icoil, intensity of the electric current in the coil
8 3000000.0                    : excitation frequency [Herz] of the coil
9 3                             : number of coils
10 0.063      0.033             : axial and radial coordinates of each coil
11 0.092      0.033             : axial and radial coordinates of each coil
12 0.121      0.033             : axial and radial coordinates of each coil
```

Take a note of the parameters that handle the boundary conditions and make sure that they match the ones set in the .geo file. Any entry that contains a bad data will trigger an error and the printing on the screen of a correct parameter file format.

The computation is then launched by the command

```
EXE\icp parameters
```