AptoFEM Placental Flow

ZC

The placental geometry creation and solver programs described here can be found at `https://github.com/ZeeCr?tab=repositories`.

1 Placenta Creation

Before running the flow solver, a placental geometry `data/` directory must be placed in the root directory of the solver. To this end, assuming an appropriate Python >= 3.9 environment has been setup with the packages within `requirements.txt` installed, we run `placenta_run.py` in the root directory of the geometry code.

As there are points of potential failure for the geometry creation (intersections of vessels due to approximations during geometry creation, unavoidable intersection of unremovable Gmsh geometry objects with placed structures, ..), this runs until it successfully creates a geometry. Once it does, it exits after creating a geometry data directory within `meshes/` with the directory name `[ID]_[COTYLEDON CONFIGURATION]_s[NO. SEPTAL]p[NO. PERIPHERAL]b[NO. BASAL]a[NO. ARTERIES]`. Inside this directory is a sub-directory named `data/` which contains all of the geometry information and meshes needed for running the flow solver. It also contains a backup of all source code at runtime.

Within the file `placenta_const.py` is a sub-header `MESH SIZE`. In this section, the mesh size in various locations of the mesh is defined. It's recommended to first test the setup described here on a very coarse mesh by increasing `DomSize`.

2 Solver

The aforementioned `data/` must be moved into the root directory of the solver. Therefore, the root directory should have the structure

- ADR_steady
- ADR_steady_markers
- data
- flow
- markers
- misc. files

Once done, to run e.g. the flow solver, a `cd flow && make && ./ns_bdf.out` will make and run the program on the created geometry. It is recommended to look at `flow_ADR_steady.py` to see how the flow parameters within `aptofem_control_file.dat` are defined. Of note are `Re, Da_reciprocal, t_scaling` which are all related to the nondimensionalisation of the program. As they are defined in the repository, these correspond to the nominal, typical characteristic scales. Further information can be found within the associated paper.

It is recommended to run the programs through the Python automation script, `flow_ADR_steady.py`, as it automatically handles the setting of parameters in the control files and file names. If not, then `README.md` details what must be changed in the code (briefly, for nominal parameter values, ensure `aptofem_run_number.dat` is 1 in the flow and ADR solvers before running them).