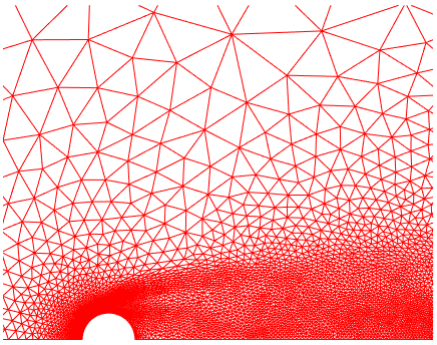

StabFem Documentation

Manual

IMFT, v0.2 May 2018

Contents

Previous Note

StabFem is a software to perform Global Stability calculations in Fluid Mechanics, which is developed for both research and education purposes. It can be downloaded at `https://github.com/erbafdavid/StabFem`. Additionally, one will previously need to install *Matlab* and *FreeFem++* on the running system. For now, *StabFem* works perfectly in *Ubuntu16.06*. On other Operating Systems, it's not sure that it will work.

This document is a collaboration of different people and is intended to explain, the best way possible, the different parts of *StabFem*. Perhaps as the reader will read it, some chapters are uncompleted or complicated to understand. The reader is invited to contribute in order to render the documentation more understandable.

Chapter 1

General View of *StabFem*

StabFem has a main directory where all the projects are located and where the mutual scripts are located too. The directory is composed by the following particular project directories:

1. ACOUSTICS_PIPES;
2. CYLINDER;
3. CYLINDER_VIV;
4. DISK_IN_PIPE.
5. etc...

The mutual directories are the followings:

1. SOURCES_FREEFEM;
2. SOURCES_MATLAB;
3. Documentation.

These directories are used by the several projects of *StabFem*.

Attention: When you do some change in the mutual directory files, we have to assure that it will work on all the projects.

1.1 General Features

- With *StabFem* we can solve forced problems, eigenvalue problems, etc...
- *StabFem* allows us to save the scripts used to generate the different data and graphs, unabling to reproduce the same results (useful for repeat figures for articles, confirming results, etc...).

1.2 What is attended from this manual document

This document presents this first chapter describing what can be found in a *StabFem* project, a chapter describing the running logic of a *StabFem* program and the main common files and a chapter for each project, describing succinctly their running features.

Chapter 2

How a StabFem project works

A *StabFem* project have a directory, for example 🗀`StabFemm>CYLINDER` where it can be found:

1. The `*.m` files corresponding to the *Matlab* scripts. Normally, a main script can be found, e.g. SCRIPT_CYLINDER_DEMO.m ;

2. A directory 🗀`StabFemm>CYLINDER > WORK` (if not, it will be created when *Matlab* script run) where it can be found the output data;

3. All the `*.edp` specific to the project and to be executed automatically by the *Matlab* interface. E.g.: The Mesh*.edp, the Macros_StabFem.edp, Param_Adaptmesh.edp, etc.

2.1 Running logic of a *StabFem* project

When one executes the main script `*.m`, both local and share scripts are executed. For a commum project, the following steps done:

1. Launch the share script SF_Start.m:

```
run('../SOURCES_MATLAB/SF_Start.m');
```

   creating the following directories as global variables and adding the *sfdir* to the *matlab* paths:

```
ff = '/PRODCOM/Ubuntu16.04/freefem/3.51/gcc-5.4-mpich_3.2/bin/
    FreeFem++'; % on IMFT network
sfdir = '~/StabFem/SOURCES_MATLAB/'; % where to find the matlab
     drivers
ffdir = '~/StabFem/SOURCES_FREEFEM/'; % where to find the
    freefem scripts
ffdatadir = './WORK/';
addpath(sfdir);
```

   It also creates the SF_Geom.edp need for *FreeFemm++*.

2. Then, a mesh and a baseflow are generated for the project with the help of the share script SF_Init.edp located in 🗀`StabFemm>SOURCES_MATLAB`,e.g.:

```
baseflow=SF_Init('Mesh_Cylinder.edp', [-40 80 40]);
```

The detailed input/output parameters are discussed in next chapter (To do...). It will execute Mesh*.edp and generate in the current path the mesh.msh, mesh.ff2m, mesh_init.msh, SF_Init.ff2m, BaseFlow_init.txt and BaseFlow_init.ff2m files.

The path StabFemm>CYLINDER > WORK>BASEFLOWS is created. This is where all the base flow for the different Reynolds numbers will be stored.

3. The baseflows for for different parameters (Re, Porosity,...) are generated by the command:

```
baseflow=SF_BaseFlow(baseflow,'Re',10);
```

executing the commom scritpt SF_BaseFlow.m. This script will read the *baseflow.mesh.problemtype* parameter and execute the corresponding Newton routing Newton*.edp located at StabFemm>SOURCES_FREE in order to generate the corresponding baseflows in their path.

4. Then, a adaptation of the mesh is made to refine and converge the results for the correct baseflow, with the following command:

```
baseflow=SF_Adapt(baseflow,'Hmax',10,'InterpError',0.005);
```

The refine parameters are written in Param_Adaptmesh.edp(why?) in the current path. Adapt_Mode.edp is executed from *ffdir* to refine the mesh. (Some files are created...explain...)

5. A mesh adaptation taking into account the eigenmode can be done: for that, first a solution have to be commuted first giving the eigenmodes. Then the mesh adaptation can be done like it was done in last step.

```
[ev,em] = SF_Stability(baseflow,'shift',0.04+0.74i,'nev',1,'
    type','D');
[baseflow,em]=SF_Adapt(baseflow,em,'Hmax',10,'InterpError'
    ,0.01);
```

Here, the eigenvalue problem has been solved with a shitf-and-invert iteration process, detail in next chapter((to do)). SF_Stability.m is once again located at *sfdir*.

6. After the former step, once wisely used, post-processing can be made. At that stage, each project have its particularities and it will be detailed in their dedicated chapters.

2.2 Create a *StabFem* project

In order to create a project in *StabFem* one has to start to code the scripts in *FreeFEM++*.

> Attention: When creating the different *.edp files, one has to pay attention on the compulsory inputs and output of the *Matlab* interface.

Then, the *Mathlab* scripts, with the previously presented style, have to be created.

> Attention: The different *.m files created in the particular directory will be used only in your project, so you can used them as you like; but, once the common files are used, they must not be changed without careful examination of the impact on the other projects.

Commonly, the following files are needed in the current file:

***FreeFEM++* file: SCRIPT_*.edp** Here, the problem is defined. See chapter (to do) for more details. It can be a eigenvalue problem, a forced problem, etc. To define the problem, see the FreeFEM++ documentation [1][1].

***FreeFEM++* file: Mesh_*.edp** File where the mesh of the problem and the convenient files are generated.

***FreeFEM++* file: Macros_StabFem.edp** Macros are a powerful tool in *FreeFEM++*. In this file all the Macros specific of the project are created. This macros will be used both by the `*.edp` scripts of one's problem and by the common scripts located at `StabFemm>SOURCES_FREEFEM`.

***Matlab* file: mains_cript.m** This script will be organise like in the previously presented style. Then, the a different treatment is given to each problem, and one can be inspired by the project already created.

[1] http://www.freefem.org/ff++/ftp/freefem++doc.pdf

Chapter 3

Mutual Matlab Files

Chapter 4

Mutual FreeFEM++ Files

Bibliography

[1] F. Hecht. New development in freefem++. *J. Numer. Math.*, 20(3-4):251–265, 2012.