The generic finite element toolbox Getfem++

The domain of numerical methods evolves very quickly. The increase of the size of computers, and the always higher requirements (in accuracy, robustness, time computation) are promising some huge developments in the future. The Generic finite element toolbox Getfem++ has been developed for making easier the conception, programming and validation of new numerical methods.

Getfem++ principles

In Getfem++, the standard concepts of calculations are clearly separated. The geometrical transformations, quadrature methods, finite elements and physical theories are completly interchangeable. It is the main feature of this toolbox.

- the meshes can be made of elements of different geometries and dimensions,
- the finite elements and quadrature methods are automatically updated after any modification of the mesh,
- the assembling procedures are generic, hence their description is independant of the space dimension, the finite elements or integration methods used.

This allows to develop codes that are particularly generic and efficient: some single parameters allow to change the space dimension, the mesh, the finite element and integration methods, and the physical model that is solved.

Present capabilities of Getfem++

A big catalogue of finite elements and quadrature methods is proposed. For instance: P_k and Q_k elements, Hermite, Argyris, HCT, XFEM, mixed methods, elements with hierarchic basis. Moreover, numerous physical models are already available: linear or large strain elasticity, Helmoltz, Stokes... For an exhaustive list, see the documentation [1]. On these features, Getfem++ is one of the most complete toolbox that exist. A large number of numerical experiments presented in international publications were carried out with Getfem++.
Getfem++ also owns the standard tools necessary for scientific computing: matrix calculus toolbox, solvers, visualization tools, mesher for simple geometries.
Getfem++ is developped in C++. Two interfaces, in Python and Matlab, enable to use all the functionalities of the toolbox without any knowledge in C++. Moreover, Getfem++ documentation is regularly updated.

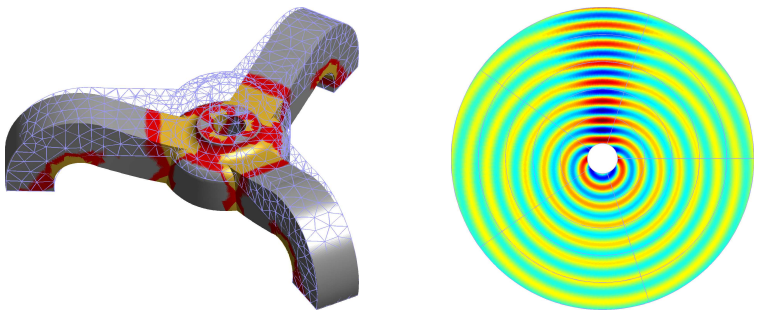

Figure 1: Left: Von Mises criteria on the surface of a tripod loaded verticaly (computed in isoparametric P_2). Right: Diffraction of a plane wave, by a cylinder.

Références

[1] J. Pommier, Y. Renard *Getfem++, an open source generic C++ library for finite element methods.* http://home.gna.fr/getfem.