Industrial CFD Process and Applications

Amir Fadel[1], Giuseppe Fontana[1]

[1]Isoil Impianti, Albano S. Alessandro, Italy

Abstract

Industrial applications of fluid mechanics can require to satisfy necessities as diverse as legal norms, optimization requirements and manufacturing constraints. Therefore a Computational Fluid Mechanics software often becomes a must in the development of new devices or the improvement of older ones. Besides the legalistic aspect (such as the European Pressure Equipment Directive), several steps are required so that a numerical simulation may be successfully used to realize an industrially viable device. First of all, there is the need for a computer-edible geometrical model; any geometrical entity must be converted into a computational domain through the meshing process. Afterwards, numerical computation requires the set-up of a solver and of a scheme to solve for highly complicated partial differential equations. Finally, an iterative process must be established to ensure the accuracy, or at least the sensibility, of the solution. COMSOL can greatly alleviates, if not entirely take care on its own, of two out of three steps; however, meshing and solving can still require an effort that amounts to the usual work any human may do in dealing with a non-reasoning machine. We illustrate such multi-step process with different examples from the industrial sector of oil&gas, dealing with fully turbulent flows within complicated geometries. In particular we show the application of COMSOL CFD Module to three different type of fully turbulent flows with Reynolds number ranging from few thousands to over 500,000. One type of geometry is relatively simple,no symmetry and no brusque change of flow direction. Another type of geometry is symmetric, and has a geometric ratio of 10^4 and rotating walls. At last we will show the most challenging geometry for COMSOL: a device with more than one brusque change of flow direction. We will show that COMSOL can almost always produce reasonable results in terms of velocity distribution and pressure drop, thus reducing the costs of prototyping.

Reference

1. Kerstin Avila et Al., The onset of turbulence in pipe flow, Science, 333, 192 (2011).

Figures used in the abstract

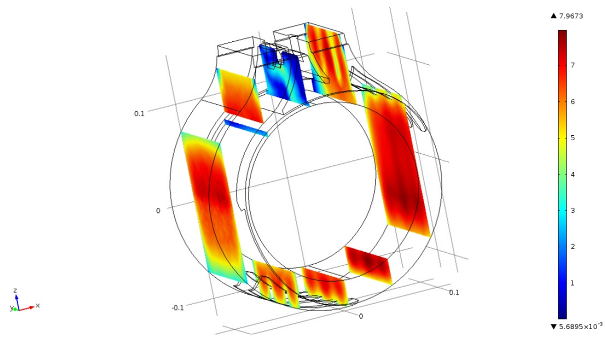

Figure 1: Velocity distribution in a flow meter. At 550 rpm, the flow does show acceptable separation and instability.

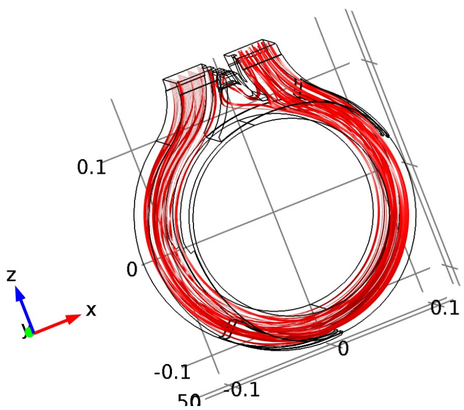

Figure 2: Streamlines in a flow meter, showing no stagnation within the measuring chamber.

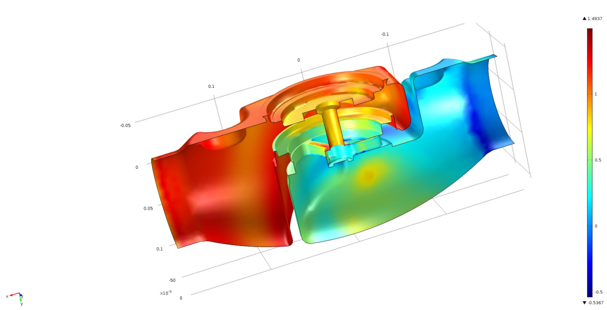

Figure 3: Pressure distribution in an open valve show improvement due to redesigning of upper portion.

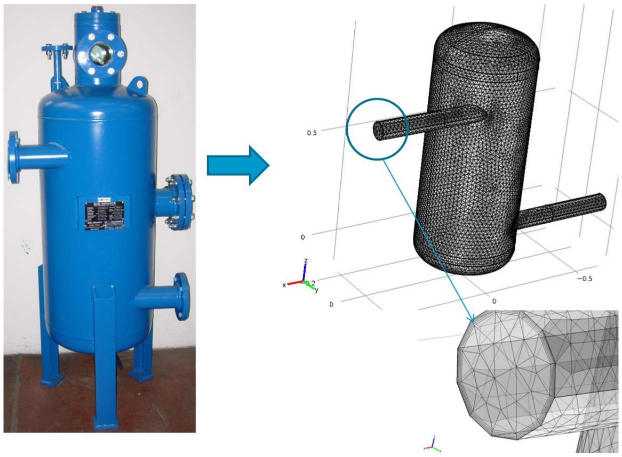

Figure 4: An example of CAD import and automatic/manual meshing with COMSOL.