Turbulent Bounded Flows for Oil & Gas Industry with COMSOL CFD Module

A. A. Fadel[1]*, G. Fontana[1], and M. Blumer[1]
[1]Mechanical R&D Department, Isoil Impianti S.p.A.
*amir.fadel@isoil-impianti.it
Isoil Impianti, via Madonna delle rose 74, 24061 Albano S. Alessandro (BG), Italy

Abstract: Industrial bounded flows must satisfy different necessities from optimization to manufacturing constraints. Therefore a CFD software is a must to develop or improve devices. Several steps are required so that a numerical simulation may be successfully used to realize a viable device. First of all, technical drawings must be read by the software; then, any geometrical entity must be converted into a computational domain through the meshing process. 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 of the solution. Comsol can greatly alleviates, if not entirely take care on its own, of two out of three steps: meshing and solving require a relatively modest effort from any experienced human operator. We illustrate different examples from Isoil Impianti S.p.A. (oil&gas), dealing with fully turbulent flows within complicated geometries.

Keywords: Computational Fluid Dynamics, turbulent flows, k-omega, bounded flows, meshing, boundary layers.

1. Introduction

Fluid dynamics deals with relative motion of different elements of a portion of fluid subject to forces, [1]. There are two types of fluid motion, laminar and turbulent. Governing equations are the same for both types of flows, the famous Navier-Stokes equations. Laminar flows are deterministic phenomena so that they are well modeled and understood; whereas turbulent flows are statistical in nature and thus difficult to represent mathematically as well as to predict, [2]. In the oil & gas industry, turbulence modeling is a top priority in order to cut back on both prototyping costs and time-to-market for new devices. Several mathematical models are available to describe turbulent phenomena; Comsol, through its CFD module, offers three models to deal with bounded turbulent flows, k-epsilon, k-omega, low Reynolds k-epsilon. At Isoil Impianti S.p.A., the mechanical research & development department has taken advantage of the capabilities of the k-omega model to solve for fully turbulent bounded flows. In this paper we want to show strengths and weaknesses of Comsol CFD module through 3 examples of fully turbulent flows: turbulent flow in a special gas separator, in a positive displacement flow-meter and turbulent flow through two coupled domains (p.d. flow-meter and strainer).

2. Examples of Comsol CFD module

Rather than a single specific application, we have decided to provide different examples in order to explore what we found to be the most important characteristics of Comsol CFD module:

- Native design with the possibility to parameterize characteristic dimensions.
- CAD import with virtual operations.
- Automatic meshing, especially for bulk mesh, whereas boundary layer mesh requires a high level of manual tweaking.
- Coupling of computational domains by means of projection operators.

2.1 Special Gas Separator

This simple examples is a static analysis of a parameterized geometry. The goal is to optimize the shape to minimize the pressure drop, while creating a vortex inside the device that may abide by International Organisation of Legal Metrology (OIML) guidelines for particle residency time in the gas separator, [3].

2.2 Positive Displacement Flow-Meter

Positive displacement flow-meters have a very complicated geometries due to very large aspect ratio (1:1,000) and moving internal parts. This example highlights the use of CAD import an native geometric operations to realize the computational domain. In this example, first we show the agreement between predictions from numerical computations and experimental results for pressure drop; then we show a new model of flow-meter developed from scratch.

2.3 Coupling of Computational Domains

In this example we show results from coupling of an air-strainer with a positive displacement flow-meter. A projection operator is used to link the dependent variables from one outlet to another inlet.

3. Use of COMSOL Multiphysics

In addition to Comsol Multiphysics (vers. 4.3), two more modules have been used, CAD import and CFD (vers. 4.3), unless stated otherwise.

The governing equations are the usual Reynolds-Averaged Navier-Stokes equations available through Comsol CFD module. Comsol built-in k-omega mathematical formulation has been used. The Reynolds number ranges from 16,000 to 1,600,000, depending on fluid viscosity.

Typical boundary conditions are inlet, outlet, wall functions. Inlet conditions have been specified to match desired flow rate, whereas outlet conditions have been defined as 0 normal stress condition. In addition to built-in impermeable walls wall-functions, also moving walls wall-functions have been used to describe a rotating motion of an inner cylinder:

$$u_i = 2\,\pi\, rpm\,(x_i - x_{0i})$$

where u_i is the i-th component of the velocity vector, x_i is the i-th spatial coordinate, $x0_i$ is the i-th coordinate of the geometrical centre of rotation, and rpm are rounds per minute.

Finally, an iterative process has been applied to check for consistency between simulation results and assumptions in the Comsol k-omega formulation of the turbulence phenomenon. Such a process is based upon an iterative control of the values of dimensionless and dimensional wall distances (δ_w^+, δ_w respectively), and a subsequent eventual boundary layer mesh refinement.

4. Results

The results have been grouped in three sub-sections, one for each example.

4.1 Special Gas Separator

Figure 1, panels a and b, show the value of δ_w^+, andδ_w respectively. The former is exactly 11,06, consistent with the approximation assumed in Comsol code. Furthermore, the dimensional wall distance appears to indicate the presence of a vortex inside the gas separator.

Figure 2, panels a and b, depicts the velocity distribution and the streamlines inside the gas separator. The streamlines show the presence of a vortex, where the lower and upper tips coincide with what predicted by the dimensional wall distance.

An analysis with Particle Tracing Module (vers. 4.2a) shows that the residence times for the particles are in line with the requirements of OIML guidelines R117-1, [3], (results not shown).

4.2 Positive Displacement Flow-Meters

Numerical analyses of existing 4” flow-meters (BM400) show an excellent agreement between measured pressure drops and predicted values for different viscous fluids (ranging from 5cSt to 200cSt), figure 3.

In figure 4, panels a and b, we show the dimensionless and dimensional wall distance for a novel model of 2” positive displacement flow-meter (SBM32). The dimensionless wall distance respects the 11,06 viscous units assumption, while the dimensional wall distance is smaller than any characteristic lengths. In figure 5 velocity and streamlines show lack of re-circulation inside the flow-meter. Reynolds number is 130,000, fluid viscosity is 0.88 cSt.

4.3 Coupling of Computational Domains

Figure 6 shows the dimensional wall distances for two computational domains, a strainer and a positive displacement flow-meter. The outlet of the strainer is projected onto the inlet of the flow-meter by means of a general projection coupling operator. Dimensional wall distance reaches local maxima whenever there is flow detachment from the wall.

Figure 7 shows the velocity distribution and streamlines inside the strainer and the flow-meter.

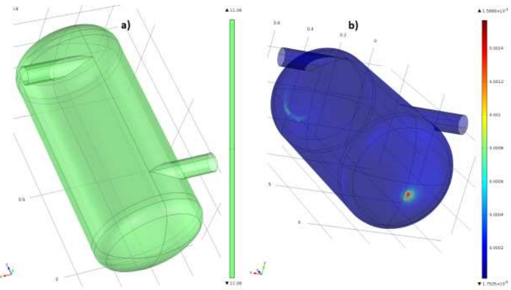

Figure 1. Panel a) shows that δ_w^+ is exactly 11.06 everywhere; panel b) shows that there are local maxima for δ_w at the top and at the bottom of the special gas separator.

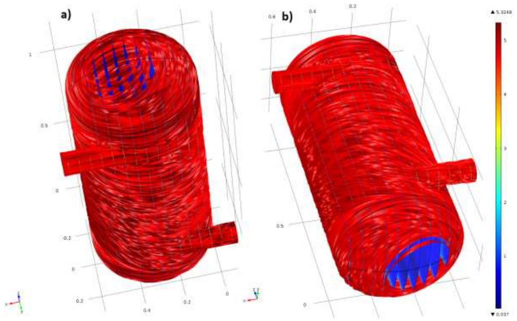

Figure 2. Velocity and streamlines inside a Special Gas Separator. Reynolds number is about 500,000. Inner vortex has upper and lower tips coinciding with local maxima for δ_w.

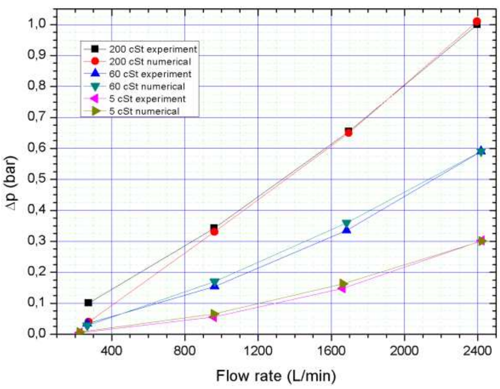

Figure 3. Experimental results show excellent agreement with numerical predictions for pressure drops. Reynolds number ranges from 100,000 to 255.

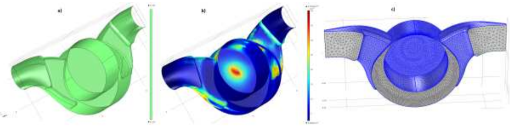

Figure 4. Panel a) shows that δ_w^+ is exactly 11.06 everywhere across the bounding surface of the computational domain. Panel b) shows that δ_w is smaller than any characteristic length. Panel c) shows the meshed domain, please note the challenging aspect ratio.

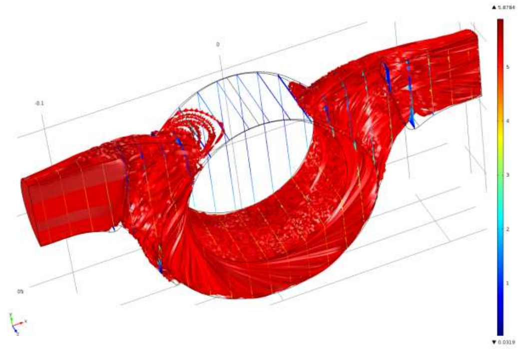

Figure 5. Velocity and streamlines inside a 2” p.d. flow-meter. No re-circulation is observed, neither in the clearance between the lateral bounding surface and the inner cylinder, nor between the inner cylinder and the top portion of the meter.

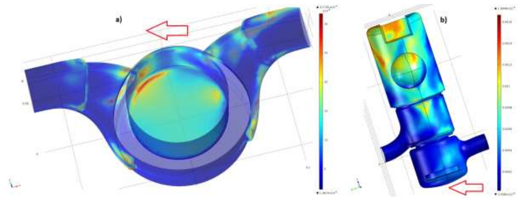

Figure 6. Dimensional distance from the wall for two coupled domains: a strainer, panel b), a p.d. flow-meter, panel a). The red arrows show flow direction.

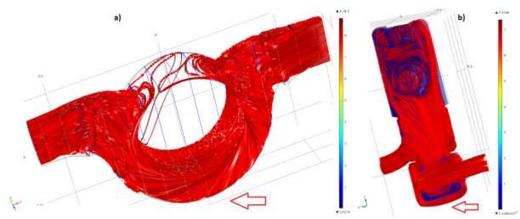

Figure 7. Velocity distribution and streamlines inside a strainer, panel b), coupled with a p.d. flow-meter, panel a). Streamlines differ from a flow-meter stand-alone simulation, figure 5, showing a little re-circulation, 3 streamlines out 1,000. Red arrows indicate flow direction. Reynolds number is about 150,000.

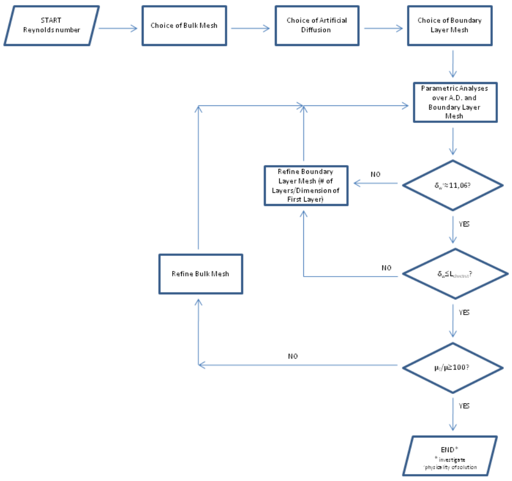

Figure 8. Simplified scheme for an iterative control over the self-consistency of the numerical solutions.

5. Discussion

The native design tool with parameterization capabilities allows for iterative simulations, and, when coupled with CAD import module, it also allows for creating complicated geometries with high aspect ratios.

The meshing tool is sufficiently automatic; while no seeding of edges is necessary even for complicated geometries, some manual tweaking is required to deal with boundary layer meshes. In particular, the dimension of the first boundary layer mesh greatly affects pressure distribution, but leaves more or less unaltered the velocity distribution. The large influence of the boundary layer mesh may be due to the assumption built in the Comsol k-omega formulation (merging of laminar sub-layer with the turbulent sub-layer, somehow skipping the intermediate sub-layer). Such dependency has led us to propose some scheme for iteratively check self-consistency of the solution. A flow-chart roughly represents the iterative process, figure 8. Problems with Comsol k-omega formulation become evident in the presence of high Reynolds impinging flows, which require special treatment for obtaining the correct pressure distribution, (results not shown).

Streamlines and particle tracing have proved to be great tools in determining fluid movements and residence times within the computational domain, allowing for large cut-down on prototyping time.

6. Conclusion

Comsol does seem to be a good tool for modeling industrial fully turbulent bounded flows.

The strengths of Comsol lie in a partially automated process that includes CAD import and, above all, meshing. A bland weakness is the need for iteratively check the self-consistency of the solutions, i.e. consistency between Comsol assumptions in its own k-omega formulation and the results.

A major weakness lies in the difficulties in modeling impinging flows.

8. References

1. Batchelor, G.K., *An Introduction to Fluid Mechanics*. 1981: Cambridge University Press.
2. Pope, S.B., *Turbulent Flows*. 2000: Cambridge University Press.
3. *Dynamic measuring systems for liquids other than water*, O.I.d.M. Legale, Editor. 2007.

9. Acknowledgements

Isoil Impianti S.p.A. must be acknowledged for consistently focusing on Research and Development, greatly improving the understanding of flow devices.

We also acknowledge all the help we have been receiving from Comsol support under several circumstances.