Using Computational Fluid Dynamics as a Compressor Design Tool

F. Fagotti
Embraco

F. C. Possamai
Embraco

Follow this and additional works at: https://docs.lib.purdue.edu/icec

https://docs.lib.purdue.edu/icec/1377

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.
Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at https://engineering.purdue.edu/Herrick/Events/orderlit.html
USING COMPUTATIONAL FLUID DYNAMICS AS A COMPRESSOR DESIGN TOOL

Fabian Fagotti, Fabricio C. Possamai
Embraco – Empresa Brasileira de Compressores SA
Rua Rui Barbosa 1020, Cx. P. 91, 89219-901 Joinville SC Brazil

ABSTRACT

Heat and mass transfer processes are of paramount importance when designing some important compressor components, specially the ones that perform flow control, noise suppression and the compression process itself. In general, pressure losses, flow pulsation and heat exchanges must be minimized to achieve high efficiencies. On the other hand, the optimization of those parameters are subject to a compromise with noise generation, component manufacturing costs and reliability, hence the bigger is the amount of information available, the higher is the probability of a successful project. In this scenario, the use of Computational Fluid Dynamics (CFD) plays an important role in the development phase, since it allows the analysis of more options, faster and at lower expenses comparing to the “prototype-and-test” approach. It also spreads possibilities of alternatives prone to be considered in a technical trade-off and anticipates problems and design constrains prior to prototyping. This paper focus on designing problems and case studies solved using CFD. Problems are described in terms of the approach used in the solution, boundary conditions, models adopted and results obtained.

NOMENCLATURE

\(A_f\) Effective flow area
\(A_w\) Effective force area
\(d\) Valve port diameter
\(F\) Force on the reed valve
\(k\) Specific heat ratio
\(m\) Mass flow rate through the valve port
\(p_u\) Flow pressure upstream the valve port
\(P_{atm}\) Atmospheric pressure
\(r\) Pressure ratio \((P_{atm} / P_u)\)
\(R\) Gas constant
\(s\) Reed valve displacement relative to its seat
\(T_u\) Flow temperature upstream the valve port
\(\Delta p_v\) Pressure difference on the valve

INTRODUCTION

Nowadays the effort required to optimize compressor efficiency has become increasingly arduous. Unlike the recent past scenario, the weights in the compromise among efficiency and other factors like noise and costs changed a lot; the rule “efficiency whatever the cost” is no more valid. Besides, the more evident solutions have already been realized, which makes necessary the use of more sophisticated tools when designing a brand new compressor or even optimizing an existing one.

Compressor design has been traditionally performed using a “trial-and-cut” approach, which imply lots of experimentation, little knowledge leaps and long lead times. In order to avoid those weaknesses, it makes necessary changing the project from the trial-and-error to a more systematic approach, augmenting the number and kind of designing tools. Computer Aided Engineering (CAE) is an important branch of those tools. Despite of the eternal skepticism about the precision of the results, in most of the cases basically a matter of proper conducted simulation, the amount of information prone to be generated using CAE tools is unquestionable large, even more when dealing with heat and mass transfer. In this case, sometimes the experimental analysis is quite difficult to carry on, due to the small room to install transducers, fast transients and strong anisotropy. This is especially true for small hermetic compressors. Moreover, experimental procedures require a prototype, which cost and time to manufacture could be unacceptably high in the early stages of a project. Therefore,
number of modifications in the original design. In version 1 a longer internal tube was introduced. Version 2 includes a nozzle in the entrance. Optimized version resulted from a completely new design. Time step corresponds to a crank angle of 10\(^\circ\), three cycles were necessary to achieve time convergence; the last one is depicted in both figures.

![Graph of Pressure vs Cycle](image1)

![Graph of Mass Flow Rate vs Cycle](image2)

**Figure (2) - Pressure fluctuation throughout a specific suction muffler design**

**Figure (3) - Inlet mass flow rate for some alternative suction muffler designs**

One can clearly see that the use of CFD helps the procedure of comparing different designs, not only in terms of efficiency (related to the mass flow rate and pressure loss), but also in terms of low frequency noise generation, analyzing the pressure pulsation at the muffler inlet. For transient analysis, the typical computational time to obtain a reasonable solution is about 100 hours using a SGI R10000 workstation.

**In-cylinder heat transfer.** It is well known the instantaneous heat transfer between the cylinder walls and the gas during the compression cycle has a strong influence on the compressor energy and mass efficiencies. Considerable heat flux occurs in the interface, from the cylinder to the gas during suction and vice-versa during compression. Moreover, for usual designs and operating conditions, the heat flux is out of phase with respect to the bulk gas to cylinder temperature difference. Experimental evaluation of the parameters involved is quite difficult to carry on using the current state of the art.

The main obstacles to simulate the problem are the strong mesh anisotropy in the top dead center and the need of a moving mesh. Due these very particular characteristics and the difficulty to deal with them using a commercial code, the approach used in this case was developing a specialist program to perform the simulation. Catto & Prata (1997) present more details about the work and a case study. The main objective was determining a simple correlation to evaluate the gas-to-wall heat transfer to be used in an overall compressor performance simulation program (see Fagotti & Prata, 1998).

**Discharge muffler optimization.** Unlike the suction muffler, the interaction between the valve and the muffler functioning is not so strong; thus steady state solutions are reasonable to some degree. In spite of discharge pressure losses do not influence compressor performance so directly comparing to the suction side, it has a considerable weight when determining stalling and starting characteristics, which are key parameters to determine the maximum motor efficiency achievable.
Computational Fluid Dynamics (CFD) becomes more and more useful according to the project targets difficulty raise.

Basically the CFD technique aims the solution of the partial differential equations of mass, momentum, energy and specimens conservation, depending on the case simultaneously or not, including a turbulence model when necessary. The spatial and time domains are divided into finite volumes or elements and intervals, respectively. Dealing with compressor design, most of the time the problems are transient, turbulent and compressible, with heat transfer and a complex geometry.

Depending on the characteristics of the problem, the time available to solve it and the accuracy required, one can chose between the use of a commercially available package or an “in-house developed” code. Table (1) shows some positive and negative aspects to be considered. Developing software requires proficiency in skills not related to the analysis itself. On the other hand, modeling and analyzing a problem via CFD always requires some level of knowledge, and the program development may be an advantageous way to achieve it.

Table 1 - Commercially available packages versus in-house CFD softwares

<table>
<thead>
<tr>
<th></th>
<th>positive</th>
<th>negative</th>
</tr>
</thead>
<tbody>
<tr>
<td>commercial</td>
<td>• no developing time</td>
<td>• black-box, weak points barely known</td>
</tr>
<tr>
<td></td>
<td>• easiness to maintain up-to-date technology</td>
<td>• high cost-to-benefit relation if the number of problems is small</td>
</tr>
<tr>
<td></td>
<td>• allows to take the benefits of competition (costs, performance, etc.)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• other users experience may be advantageously used</td>
<td></td>
</tr>
<tr>
<td>in-house</td>
<td>• deep knowledge of the software deficiencies</td>
<td>• time consuming development</td>
</tr>
<tr>
<td></td>
<td>• developing is the best training</td>
<td>• requires a somewhat dedicated staff, which vies with a dedicated and usually greater body</td>
</tr>
<tr>
<td></td>
<td>• customized solutions</td>
<td>• “out-of-the-target” issues (programming, interfacing, debugging, model validation)</td>
</tr>
</tbody>
</table>

Of course, in the lack of a deep knowledge on the basics, none of the approaches is profitable. Commercial packages tend to be robust in terms of convergence, what means sometimes one can obtain solutions even to ill-posed problem. The quality of the solution is always proportional to the quality of modeling and the user is in charge of assuring a proper model. Some level of knowledge is required on heat and mass transfer, programming and CFD techniques themselves in order to delivery a proper model to the software. Setting up boundary conditions requires both theoretical and practical experience, in order to avoid including in the model unnecessary characteristics, which increases the computational time adding no quality to the solution.

APPLICATIONS

Several phenomena that take place in the compressor are prone to be analyzed using the CFD technique. The following problems are some of the cases analyzed by the authors to different extent.

Pressure losses and gas pulsation in suction mufflers. Most of the efforts demanded when designing a suction muffler is driven towards the compromise between noise and efficiency. Its most important are attenuating the noise generated by the suction valve and conducting the working fluid to the suction port at a minimum temperature increase and pressure loss. The CFD technique is valuable to evaluate the last two parameters; for noise attenuation there is a route different to some degree. In spite of the solution of the full Navier-Stokes equation gives complete information about the pressure field, which is basically the main parameter analyzed when dealing with noise, in order to capture high frequencies one should use very small time steps. This implies a required precision much higher than the usual to analyze flows, leading to almost impracticable computational
times considering the computers available nowadays. Thus, for noise purposes a simplified version of the momentum conservation equation is used. Obviously, both noise and flow problems must be considered interactively. In this section, the muffler performance is discussed only in terms of energy parameters (temperature increase and pressure losses).

Considering the strong coupling between suction muffler and valve, in order to obtain reliable solutions one must consider the flow as transient. Since temperature and gas compressibility are important issues, the energy conservation equation can not be neglected. Furthermore, as velocities vary a lot, turbulence and transition models fit must be taken into account. With all those aspects involved, strict geometry modeling is very important; geometric details located far away from the main flow can be omitted to diminish computational and modeling efforts.

Table (2) compares steady state and transient results for a muffler with the very same boundary conditions, that is, the boundary conditions for the transient solution at a specific time step has been used to evaluate a steady solution. Parameters imposed were the mass flow rate at the outlet and the mean inlet pressure. It is clear that results differ a lot, what means the steady state analysis may lead to somewhat wrong conclusions. In this case the time instant chosen for analysis is near the valve closing; if a peak condition is evaluated, of course the differences should amplify.

Table (2) – Suction muffler steady-state and transient solutions

<table>
<thead>
<tr>
<th>solution</th>
<th>inlet mass flow rate (g/s)</th>
<th>pressure loss (Pa)</th>
<th>maximum velocity (m/s)</th>
<th>maximum/ minimum density</th>
<th>maximum turbulent viscosity (kg/ms)</th>
</tr>
</thead>
<tbody>
<tr>
<td>steady-state</td>
<td>3.73x10⁻⁴</td>
<td>1570</td>
<td>45.2</td>
<td>1.03</td>
<td>3.25x10⁻⁴</td>
</tr>
<tr>
<td>transient</td>
<td>7.97x10⁻⁴</td>
<td>6730</td>
<td>79.9</td>
<td>1.11</td>
<td>5.72x10⁻⁴</td>
</tr>
</tbody>
</table>

The mesh used is depicted in figure (1); it has been divided into two parts just for an easier visualization. Note the connecting tube shown in both pictures. Fluent was the CFD code used.

Figure (1) – Muffler grid used in the steady-state versus transient analysis; (a): first volume, (b): second volume

Figure (2) compares the average pressure pulsation in the muffler inlet, outlet and volumes (see figure 1 for reference). The low frequency attenuation becomes evident. Figure (3) shows the mass flow rate evaluated for a
In a specific discharge muffler design, the main element in gas pulsation dissipation is a restriction between the two existing volumes. This restriction implies higher torques in high mass flow rate operating conditions. The objective of the present analysis was to minimize the pressure losses, enlarging the hole that represents the flow restriction between the first and second volumes. In order to avoid negative effects on noise, additional volumes were included to attenuate the gas pulsation. Figure (4) shows the pressure contours evaluated for the original design and three proposed solutions aiming pressures loss reduction. Note that reference pressures are different from one case to another, so only ranges are comparable. All solutions were obtained using the Fluent CFD package.

Figure (4) – Pressure drop for alternative discharge muffler designs (1 – top left: original, small hole communicating first and second volumes; 2 – top right: enlarged hole and additional volume to avoid pulsation increase; 3 – bottom left: smaller second volume and repositioned third volume; 4 – bottom right: no restriction between volumes and alternative volume construction)
A known mass flow rate was imposed at the inlet. As discharge tube is relatively long, a constant pressure was considered at the outlet. Turbulence was taken into account using a $\kappa$-$\epsilon$ renormalization model, since standard $\kappa$-$\epsilon$ model leads to underestimated pressure losses. Table (3) presents the head losses evaluated for each design.

<table>
<thead>
<tr>
<th>case</th>
<th>1 (original)</th>
<th>2 (enlarged hole + 1 volume)</th>
<th>3 (enlarged hole, smaller volume 2)</th>
<th>4 (no significant restriction)</th>
</tr>
</thead>
<tbody>
<tr>
<td>pressure loss (MPa)</td>
<td>1.72</td>
<td>1.49</td>
<td>1.29</td>
<td>1.12</td>
</tr>
</tbody>
</table>

The validation has been done for the first case. Simulation results (mainly pressures) differ in less than 10% relative to the experimental ones. The differences observed make evident the CFD usefulness, since the whole simulation process (four cases) cut the evaluation time to about 1/4 comparing to the traditional prototyping-and-test procedure.

Piston secondary movement & journal bearing simulation. During the piston motion it bears the forces due to the compressed gas, the connecting rod action, the hydrodynamic pressure developed in the oil film, the friction (viscous and metallic) and its own inertia. As those forces are unbalanced, piston rotates and translates relative to the bore axis. This secondary movement can induce wear and increase both gas leakage and friction loss.

Journal bearings form a system highly dependent from each other, which can be discretized at different levels. The basic model simulates each bearing without considering its interaction to the other ones. The top-level model considers the interaction between the shaft main and secondary bearings, the eccentric and their dependency on the connecting rod load and electromagnetic loads. The solution of this problem consists in solving both a hydrodynamic problem (via CFD) and the piston dynamics in the first case and the system dynamics in the second one. The hydrodynamic problem is laminar, two-dimensional and cavitation modeling is an important issue. Due to these characteristics, a commercial code could be difficult to manage (due to the concomitant solution) and time consuming (due to the relative easiness on customizing the Reynolds equation solution). Therefore, the decision was to develop a specific code. For details, see Manke et al. (1993). The software evaluates mainly the oil mass flow rate, the viscous dissipation and the shaft trajectory. Minimum oil film thickness is a parameter that well correlates with wear. Examples of possible analysis are including undercuts (to minimize friction losses), changing dimensional characteristics, evaluating alternative oils, analyzing different operating conditions and rotational speeds, predicting effects of form and position errors and optimizing the pumping oil shaft groove.

The piston problem is quite similar in terms of modeling, although the dynamic equations are somewhat particular. The complete model is described in Prata et al. (1998). Similar to the bearing simulation, the input data are the geometric characteristics, the oil properties and the operating conditions. The program results are the pressure field, the piston eccentricity during the movement, leakage and friction parameters and the forces involved. Taking the eccentricities into account, one can predict whether a design is wear prone or not, for example, correlating a minimum oil film thickness with a known wear fault. Compressor performance can also be optimized, minimizing both the friction loss and leakage.

Evaluating valves effective flow and force areas. In compressor reed valves the gas flow is the driver mechanism that causes the valve movement. Therefore, it is very important to understand how the gas flows and its connection with the geometrical parameters. Usually those characteristics are evaluated through the experimental measurement of the effective flow and force areas for specific displacements between the reed valve and the valve seat, according to the following equations.

\[
A_f = \frac{\dot{m}}{P_u \sqrt{(k-1)RT_u} \sqrt{r^{2/k} - r^{(k+1)/k}}} \\
A_w = \frac{F}{\Delta p_v}
\]
This procedure allows to compare different valve designs and to choose the best one, but do not evinces the optimized one in terms of fluid flow. The CFD approach allows to visualize the flow shape, regions with recirculations, localized pressure losses and flow curvature, which significantly helps the designer to optimize valve and port shapes.

Figure (5) shows the mesh and figure (6) depicts the velocity vectors for a valve displacement of 0.009m.

![Mesh and Velocity Vectors](image)

Figure (5) – Mesh used in the suction valve simulation  Figure (6) – Velocity vectors evaluated for the suction valve simulation

Figure (7) illustrates a comparison between numerical and experimental results of effective flow areas considering four different suction valve port diameters.

![Comparison Graph](image)

Figure (7) – Comparison between numerical and experimental effective flow area for different displacements and port diameters.
The effective areas were evaluated by solving the mass and momentum conservation equations for a steady state, isothermal, incompressible and turbulent flow. The unstructured mesh was created from a CAD solid model using tetrahedric elements. The pressure-velocity coupling was solved applying the SIMPLEC methodology. The physical properties on the control volume faces were estimated by the QUICK scheme and the renormalized $\kappa-\epsilon$ model was used to calculate turbulence. The boundary conditions were velocity inlet at the entrance and pressure outlet in the opposite side. The numerical simulations were carried out using the CFD program Fluent.

It can be seen a good agreement between the numerical and the experimental data, strengthening the powerfulness of CFD application in this case. Adjusted curves are within a 20% confidence interval comparing to experimental data and in all cases the general trends are well predicted. Moreover, paying attention to pressure contours and velocity vectors anticipates design weaknesses.

CONCLUSIONS

CFD has been applied in the solution of a number problems found in compressor design. The technique is suitable for application in virtually any problem found in the project involving heat and mass transfer. Present computational capabilities allow dealing with characteristics that would make the analysis impossible in the near past, e.g., complex geometries, transients, turbulent and compressible flows. The validation of usual problems that occur in the compressor design are mostly feasible considering mean values of the parameter analyzed, since transients and room to install transducers are concerns. The choice on using "in-house" or commercial codes depends largely on the kind of problem, schedule and available skills. Whatever the approach used, the success depends greatly on the user skills. Proper boundary conditions setting, correct choice of time step and element size variation throughout the domain, congruent domain extension and model detailing are key factors to achieve an answer that agrees well with experimental values at reasonable time and computational expenses. Besides the savings in terms of time and costs comparing to testing several prototypes, simulation results overcomes those obtainable via an experimental evaluation regarding the quality and amount of information, in spite of it does not exempts the designer of an experimental validation and conclusions corroboration.

REFERENCES