2000

CFD a Viable Engineering Tool for Compressor Valve Design or Just a Toy?

F. Ottitsch
Hoerbiger Corporation of America

P. Scarpinato
Hoerbiger Corporation of America

CFD A VIABLE ENGINEERING TOOL FOR COMPRESSOR VALVE DESIGN OR JUST A TOY?

Author: Dr. Franz Ottitsch
Paul Scarpinato
Hoerbiger Corporation of America
3350 Gateway Drive
33069 Pompano Beach, Fl

ABSTRACT

The single most important technical feature of a compressor valve is its available effective valve area. This area depends on the primary design of the sealing element (type, spacing...) and the lift at which the valve is used.

The traditional method of developing a new design is by means of an analysis of the available lift area and an estimation of the effective valve area by compensating for flow losses through loss coefficients which are known from experience. Any such new design has to be evaluated in a prototype development phase and the actual available valve area has to be verified by means of a measurement in a wind tunnel. If the testing reveals undesirable results the design has to be reevaluated, a new prototype has to be built and the flow testing has to be repeated.

Today's demands for reduced time to markets of new products triggered a need for a faster method of this development process. One obvious approach for achieving this goal is by means of a numeric experiment thus solving the governing fluid mechanic equations numerically instead of building an actual prototype.

The standard equations describing the three-dimensional stationary flow situation through a compressor valve were already derived in the 18th century. However, it is known that for most practical applications these equations show a chaotic behavior, which is usually referred to by the term "turbulence". Several models exist which try to account for this phenomena. Nevertheless none of these models is able to perfectly simulate a flow situation in which turbulence plays a major role.

The purpose of this paper is to compare the results of a "state of the art" commercially available CFD software with the results of wind tunnel testing for several common valve types (reed-, poppet-, ring- and plate valve). The comparison also takes into account the different available turbulence models and shows how these models change the outcome of the calculations. Therefore a recommendation will be given on the usefulness of this type of tool for the valve design process.

NOMENCLATURE

Q ...	Volume flow
Δp ...	pressure loss across valve
Φ ...	effective valve area
ρ ...	gas density

INTRODUCTION

The process for the market release of a new compressor valve usually consists of the prototype phase, in which different design principles are investigated and the field test phase, in which the life time requirements have to be met. The prototype phase consists of finding a compromise between the available effective valve area Bauer [1] and Lin [3], the required spring force for the closing of the sealing element, the allowable valve losses, the allowable clearance volume, the required MTBF and the required operating conditions.

In order to define the available effective valve area an iterative process has to be carried out during which this area is defined through initial flow assumptions, prototype manufacturing and design refinements until the required effective area is achieved.

The initial expected effective valve area is usually found through the evaluation of the available lift area and the anticipation of loss coefficients which are known from experience and or prior measurements of similar configurations.

For every design option a prototype has to be built to evaluate the actual effective valve area. The closer a new design is to an existing one the smaller is the difference between anticipated and actual effective area. For a totally new valve approach the accuracy of the traditional approach is usually +/-10% accurate and therefore it is not uncommon that it takes several production and evaluation cycles for a new prototype design.

Cyklis [2] has shown that the CFD method gives accurate results for a flat ring valve design. He solved the steady state two-dimensional equations for a compressible flow. Deschamps, Ferreira, and Prata, [4] have solved the time averaged Navier-Stokes Equations for a radial diffuser with axial feeding and have proved that the results are in good agreement with experimental results. They used the RNG k-e turbulence model Orzag et. al [8] and report that this model improves the prediction of separation regimes.

Due to these promising results a project has been undertaken with the goal to evaluate whether a commercially available CFD computer program can be used as an engineering tool to reduce the lead time for a new prototype. The automotive and airplane industry has been using these tools for decades. However the amount of necessary man- and computing power has until recently made the CFD technology unattractive for most of the other industries. Primary to this project a comparison of the available low end CFD packages was undertaken. The requirements for the CFD program were defined as follows:

- Low end CFD package with an interface to standard 3D CAD system (SAT)
- Program must run on a High End PC
- Results of the effective flow area of a simple sample valve has to be achievable within a reasonable amount of time

The program which offered the best compromise between price and usability was purchased.

VALVE DESIGN - ASSUMPTIONS

The flow through a compressor valve during the suction intake phase or discharge phase can be described as a three dimensional, time dependent, turbulent compressible flow. During the design phase where the effective area of a new valve is defined, the compressibility of the fluid and the instantaneous flow are usually neglected Bauer [1]. (An accurate modeling of these effect is beyond today's standard know-how and standard capabilities.) Thus the main criteria for judging the aerodynamic efficiency of a new design is the effective valve area (Φ). This area is found by measuring the differential pressure before and after a valve at a given flow rate (in the turbulent regime) in a wind tunnel. The definition of this area (Φ) is as follows:

$$\Phi := Q \cdot \sqrt{\frac{\rho}{2 \cdot \Delta p}} \qquad (1)$$

In order to replace the wind tunnel experiment by a numerical experiment an adequate equivalent has to be used. The experiment in the wind tunnel can be simulated through the model of a steady state, turbulent incompressible flow Zierep [6]. Hence the Navier Stokes equations have to be time averaged and solved. Thus the dependent variables (flow, pressure) are represented through a mean value and a fluctuating value. This representation is substituted into the governing Navier-Stokes Equations and the equations themselves are averaged over time [5] and [7]. This averaging process produces extra terms in the momentum and energy equations which are combinations of the fluctuating quantities. These extra terms are referred to as Reynolds stress terms. Thus one arrives at five equations with 14 unknown variables. The resulting closure problem has to be solved through additional modeling assumptions. A common assumption (isotropic effects of turbulence) is to combine the Reynolds stress terms into two numbers the so called eddy viscosity and the eddy conductivity. This then leaves only the eddy viscosity and conductivity to be determined which can be done with additional models like the k-ε model in which the transport of the turbulent kinetic energy is being modelled. Another approach for the closure problem is called the RNG two-equation model in which the momentum equations are transformed to wave-number space and renormalization group theory is used to derive the equations for calculating eddy-viscosity.

For the scope of the present paper it was decided to utilize the different models (k-ε, RNG) which are available from the CFD-package being used.

SCOPE OF PRESENT COMPARISON

None of the above mentioned available closure/turbulence models correctly describe the flow separation regime for a general geometry. This is specifically true if the separation occurs due to boundary layer effects (which are to a large extent influenced by the local turbulence level close to this layer) and not due to the geometric constraints of the flow boundary (i. e. sharp corners...). This is also the primary objection against the utilization of a CFD package. Thus for the comparison of the results between measurements and calculations it seems advisable to distinguish between designs where the flow separation point is clearly defined through the geometry and designs where the separation is influenced through boundary layer effects. With these designs it is possible to gain a part of the kinetic energy of the flow back after the gas passed the valve sealing element and to transform this kinetic energy into local pressure energy.

Therefore in order to evaluate the CFD technology different valve concepts were looked at by comparing the results of a CFD calculation with the results of a flow test. As samples for valve designs with sharp corners at which the flow separation point is well defined a

- ❑ single Reed valve and
- ❑ a traditional plate valve

were analyzed.

As examples for valve designs with smooth geometries at which flow separation is influenced by boundary layer effects

- ❑ a poppet valve
- ❑ and several different profiled ring valves

were taken into considerations.

All of the numerical models were set up and tested in the same way. A three dimensional drawing of the valve was drafted on a CAD system and then imported into the CFD software using the ACIS file protocol. The numeric model of the valve was set up with the sealing element in the fully open position. Before and after the valve a cylindrical tube defined the edge of the flow geometry. The size of this front and back flow regime was defined through the usual requirements for a stabilized flow. The pressure drop across the valve was measured at the end surface of the tube (average of all nodes). The average flow velocity at the exit surface was utilized to verify the overall conversation of mass. The correct amount of the grid points was verified by confirming the independence of the result on the number of grid points.

Within this paper all results are presented as the percentage difference between the calculated effective area and the actual measured value. A negative number defines that the calculated values for the effective valve area are below the values of the measured one.

Valve Designs with sharp corners

Single Reed Valve

As the uttermost simple example for the flow through a valve a single reed element was looked at. The deflection of the reed was defined by the shape of the guard. Fig.1 shows a cut-away of the velocity profile through the valve. The comparison between calculated and measured values revealed an accuracy of the calculation of –4.2%.

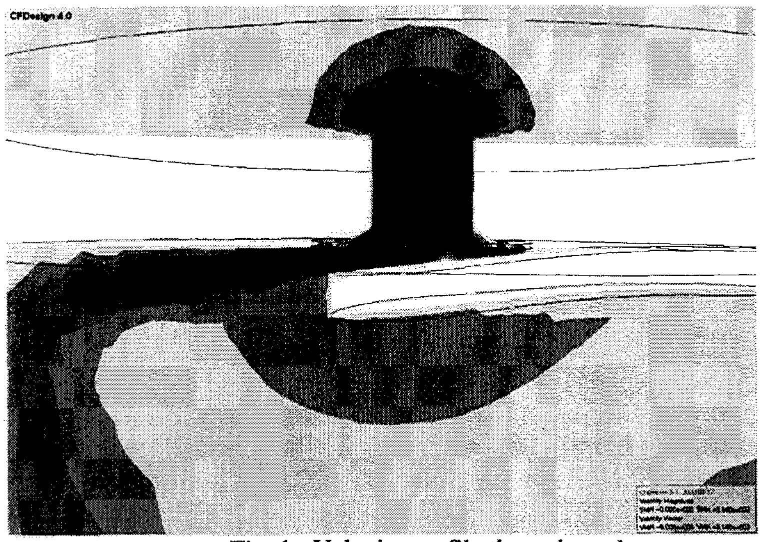

Fig. 1: Velocity profile through reed.

Plate Valve

In order to determine the possible accuracy of a traditional plate valve, a Hoerbiger type 147CT valve was modeled. The 147CT model was first calculated using the k-ε turbulence modeling method for 1748 iterations. This resulted in an error of –5.2% between calculated and measured value. The calculation was then continued to 2081 iterations using the RNG turbulence modeling method. After the additional 333 iterations the error was reduced to –2.4%.

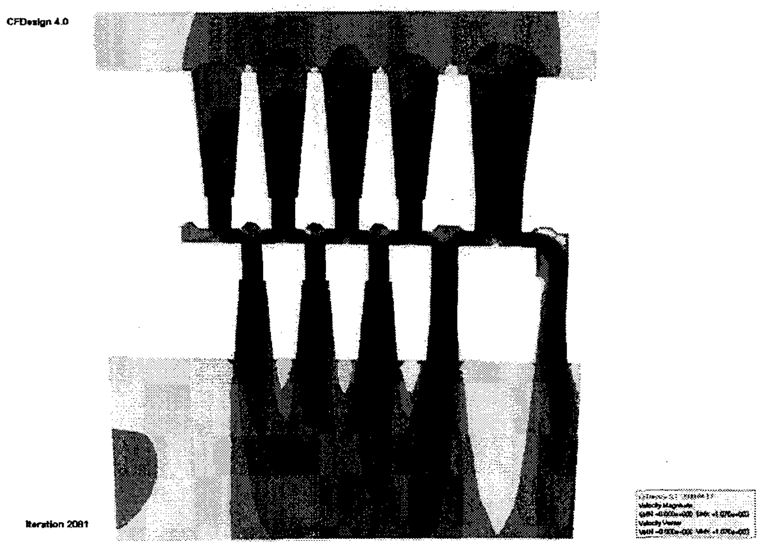

Fig. 2: Velocity profile through plate valve.

Valve Designs with smooth geometry

Poppet Valve

As an example for a valve with a smooth geometry a poppet valve was designed with seven poppets (a center poppet with six poppets surrounding the center). The poppet valve, like the 147CT valve, was calculated both with the k-ε and RNG turbulence modeling methods. The first calculation was run for 1711 iterations to give an error of –9.2%. The model was then continued for another 2101 iterations to give an error of –0.8%. The RNG modeling method did require a considerable larger amount of iterations, however the overall accuracy was surprising. The CFD did reveal an uneven flow situation at all of the outer guard ports which are fed through only one poppet, see Fig. 3.

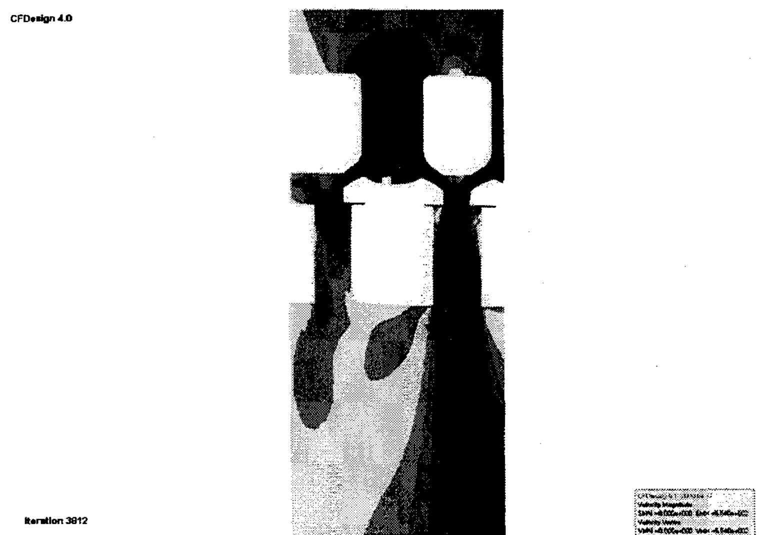

Fig. 3: Velocity profile through poppet valve.

Profiled ring valves

In order to test the possibility of predicting the valve performance for a new design using the CFD package, prototypes of three different ring designs were manufactured and flow tested. The difference in the designs were caused by design restrictions of the other valve components. The tested results were then compared to the CFD results. The accuracy of the three different designs was -1.2%, +4.0% and –2.0%. The calculated results were all very close to the measured values, however the prediction of the most efficient valve was reversed. The CFD method was not capable of predicting the correct ranking between two different design whose effective flow area differed by 0.8%.

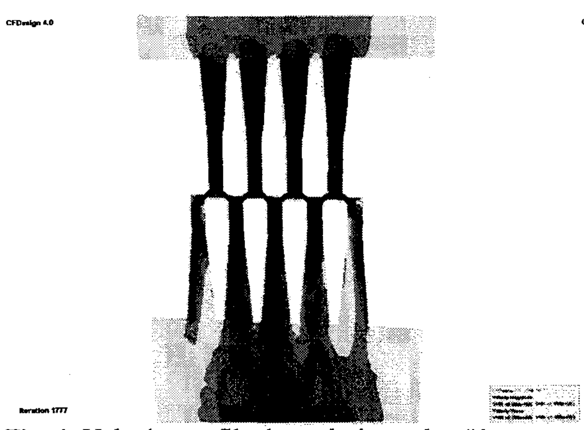

Fig. 4: Velocity profile through ring valve #1.

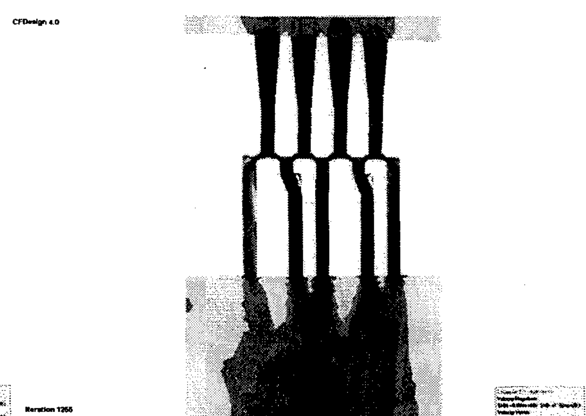

Fig. 5: Velocity profile through ring valve #2.

Summary

The following table gives a summary of the different designs which had been looked at and the difference between calculated and measured effective area.

Description of Test	percentage of error of the effective valve area
Single Reed Valve	-4.2%
Plate Valve k-ε turbulence model	-5.2%
Plate Valve RNG turbulence model	-2.3%
Poppet Valve k-ε turbulence model	-9.2%
Poppet Valve RNG turbulence model	-0.8%
Profiled ring valves	-1.2%, +4.0%, -2.0%

It is interesting to note that the values for the effective valve area of all the CFD calculation results were slightly below the measured valves. The ring valve # 2 in which the flow situation is highly asymmetric was the only one were the results of the CFD calculation anticipated a larger effective area. It could also be proven that the RNG turbulence model is the better choice than the traditional k-ε model for modeling the turbulent flow through valves (see also Deschamps, Ferrara and Prata [4]).

CONCLUSIONS

Today's commercially available CFD packages can predict the effective flow area of a valve with a surprising degree of accuracy. The computer CPU and RAM requirements are within the scope of modern PC technology. Distinguishing between different design alternatives with minute differences in the effective valve area is however questionable. Whether or not an engineer has to be able to take the correct choice between two different designs options whose effective valve area differs by only 0.8% will depend on the requirements for specific new designs.

REFERENCES

[1] Friedrich Bauer, Valve Losses in reciprocating compressors, 1988 International Compressor Engineering Conference at Purdue, pp. 263-270

[2] Piotr Cyklis, CFD Simulation of the flow through reciprocating compressor self-acting valves, 1994 International Compressor Engineering Conference at Purdue, pp. 427-432

[3] M. Lin, S. L. Pan, P. C. Shu, Study of the effective flow area of compressor plate valve, 1994 International Compressor Engineering Conference at Purdue, pp. 181-186

[4] C. J. Deschamps, R. T. S. Ferreira, A. T. Prata, Turbulent flow through valves of reciprocating compressors, 1996 Internation Compressor Engineering Conference at Purdue, pp. 377-382

[5] CFD Design Solver Technical Reference Version 3, Blue Ridge Numerics

[6] Juergen Zierep, Grundzuege der Stroemungslehre, ISBN 3-7650-2038-9

[7] J. C. Tannehill, D. A. Anderson, R. H. Pletcher, Computational Fluid Mechanics and Heat Transfer 2nd Edition ISBN 1-56032-046-X

[8] Orzag S. A., Yakhot, V. Flannery, W. S. Boysan, F. Choudury, D. Marusewski, J. Patel, 1993 Renormalization Group Modeling and Turbulence Simulations. So, R. M. C., Speziale C. G., Launder B. E. (editors), Near-wall turbulent flows. Elsevier Publications