Numerical analysis of the turbulent fluid flow through valves. Geometrical aspects influence at different positions

This content has been downloaded from IOPscience. Please scroll down to see the full text.

(http://iopscience.iop.org/1757-899X/90/1/012026)

View the table of contents for this issue, or go to the journal homepage for more

Download details:

IP Address: 147.83.83.71
This content was downloaded on 12/04/2016 at 17:17

Please note that terms and conditions apply.
Numerical analysis of the turbulent fluid flow through valves. Geometrical aspects influence at different positions.

J. Rigola¹, D. Aljure¹, O. Lehmkuhl¹,², C.D. Pérez-Segarra¹ and A. Oliva¹

¹Heat and Mass Transfer Technological Center (CTTC) Universitat Politècnica de Catalunya
BARCELONA Tech (UPC) C/ Colom, 11 08222 Terrassa (Barcelona), Spain
²Termo Fluids, S.L. C/ Magí Colet, 0 08204 Sabadell (Barcelona), Spain
E-mail: cttc@cttc.upc.edu

Abstract. The aim of this paper is to carry out a group of numerical experiments over the fluid flow through a valve reed, using the CFD&HT code TermoFluids, an unstructured and parallel object-oriented CFD code for accurate and reliable solving of industrial flows. Turbulent flow and its solution is a very complex problem due to there is a non-linear interaction between viscous and inertial effects further complicated by their rotational nature, together with the three-dimensionality inherent in these types of flow and the non-steady state solutions. In this work, different meshes, geometrical conditions and LES turbulence models (WALE, VMS, QR and SIGMA) are tested and results compared. On the other hand, the fluid flow boundary conditions are obtained by means of the numerical simulation model of hermetic reciprocating compressors tool, NEST-compressor code. The numerical results presented are based on a specific geometry, where the valve gap opening percentage is 11% of hole diameter and Reynolds numbers given by the one-dimensional model is $4.22 \times 10^5$, with density meshes of approximately 8 million CVs. Geometrical aspects related with the orifice’s shape and its influence on fluid flow behaviour and pressure drop are analysed in detail, furthermore, flow results for different valve openings are also studied.

1. Introduction

The radial outflow between two coaxial disks is technologically important for different applications (radial diffusers, air bearings, valve reeds, etc.). In all these cases the flow is quite complex, presenting a pressure gradient that may be either positive or negative depending on radial location. Furthermore, turbulent flow is often encountered, thus, turbulence models are critical on the numerical resolution of this technological relevant application.

The numerical results are obtained using the three dimensional, parallel, unstructured and object oriented code TermoFluids [1] using LES models, a high Reynolds number (close to real working conditions) and turbulent inlet flow. In fact, the present paper is an updated version of previous works [3] and [4] that used extruded meshes, low Reynolds numbers and did not use low Mach models[5].

Experimental ([6] and [7]) and numerical ([10] and [11]) have been performed setting a basis to understand the physics underlying in the flow. First attempts of numerical simulation of turbulent flows by means of RANS $k - \varepsilon$ and $k - \omega$ models were carried out under low Re numbers ([12] and [13]). More recently, some papers presents numerical results based on LES turbulent models for low Reynolds number $2.5 \times 10^4$ [14], or RANS turbulent models for higher Reynolds numbers $1.6 \times 10^6$ [15]. Finally, two-dimensional numerical cases that were experimental validated under specific experimental apparatus, although for very low Reynolds numbers, were presented in [16].

From a compressor study viewpoint the fluid flow is composed by two different physics (entrance flow through a channel and a free jet through a surface)(see fig 1a). In that sense, the present paper is focused on the numerical simulation model of the fluid flow through the valve reeds, considering a simplified geometry of an axial hole plus a radial diffuser. The methodology proposed by [17], [18] and [19] is based on two parameters: i) effective flow area $(K_A)$ that relates the actual mass flow...
rate with an ideal mass flow rate per unit flow area (assuming isentropic contraction process) defined as 
\[ m = (KA)_{e} \rho u \sqrt{2 \rho u (p_u - p_d)} \]  
\[ \text{and ii) the effective force area } A_f \text{ defined as ratio of the actual net force on the valve and the force obtained assuming a constant pressure drop distribution,} \]
\[ F = (p_u - p_d)A_f \], the most usual method for valve analysis and compressor design [2].

2. Definition of the case
The geometries to be considered are: geometry A (straight) where the hole is cylindrically shaped with constant diameter, and geometry B (cone) where the hole is drawn in a conical way linearly enlarging the diamter. Both cases are fitted with a top plate acting as the valve disc.

Fig. 1 shows the geometrical features and computational domain to be studied, where
\[ D = 1.5d, \quad s = 0.11d, \quad h = 0.24d, \quad \alpha = 20^\circ \text{ and } l = 15d. \]

Flow inlet is done through the bottom boundary such that
\[ Re = \frac{W_{ref} d}{\nu} = 4.22 \times 10^5 \text{, result obtained from NEST-compressor simulations. } \]

3. Mathematical formulation
3.1. Governing equations and numerical method
In order to study the flow, the filtered incompressible Navier-Stokes equations are solved:
\[ \frac{\partial \tilde{u}_i}{\partial t} + \frac{\partial (\tilde{u}_i \tilde{u}_j)}{\partial x_j} - \nu \frac{\partial^2 \tilde{u}_i}{\partial x_i \partial x_j} + \rho \frac{\partial \tilde{p}}{\partial x_i} = -\frac{\partial \tau_{ij}}{\partial x_j} \]
\[ \frac{\partial \tilde{u}_i}{\partial x_i} = 0 \]

where \( \tilde{u}_i \) is the three-dimensional filtered velocity, \( \tilde{p} \) is the filtered pressure field, \( \nu \) stands for the kinematic viscosity, \( \rho \) for the density of the fluid and \( \tau_{ij} \) is the SGS stress tensor defined as
\[ -2\nu_s \delta_{ij} + 1/3 \tau_{ij} \delta_{ij}. \]

Governing equations are discretized on a collocated unstructured mesh by means of finite volume techniques. A second-order conservative scheme is used for the spatial discretization [24]. Such schemes preserve the symmetry properties of the continuous differential operators and ensure both, stability and conservation of the kinetic-energy balance. The velocity-pressure coupling is solved by means of a fractional-step algorithm. The temporal discretization for the convective, diffusive and derivative parts
of equation (1) was made using a second order self-adaptive scheme [27]; whereas a back-ward Euler scheme was used for the pressure gradient. For more details about the discretization the reader is referred to [28] and [29].

3.2. Large eddy simulation turbulence models

In LES the spatial filter applied to the Navier Stokes equations separates large and small scales. This operation introduces new variables into the system of equations, the sub grid stresses (SGS). In order to solve the equations this term must be properly modelled. Four closure models are used in the present paper, WALE, VMS, SIGMA and QR, presented in [30]. The WALE model was proposed by [31] based on the square of the velocity gradient tensor. This SGS model accounts for the effects of the strain and rotation rates, as well as, a $y^+$ near wall scaling for the eddy viscosity. The variational multiscale method was formulated for the Smagorinsky model on a spectral domain by [32]. Later, this method was extended to the classical filtering approach in LES. The VMS method consists on applying a spatial test filter to the already filtered scales to divide the resolved scales of motion into large and small. The later are used to model the turbulent viscosity. QR model was proposed by [33] responding to the question of damping sub filter scales properly. It is a SGS model based on the invariants of the rate-of-strain tensor. Finally, SIGMA model, proposed by [34], is a subgrid-scale model derived from the analysis of the singular values of the resolved velocity gradient tensor.

4. Numerical results

4.1. Mesh and time integration analysis

Hybrid tetra-prism unstructured meshes are constructed to discretize the equations. A fine near wall mesh is necessary to correctly solve the boundary layer and the physical phenomena associated with this zone. A prism layer is appropriate in this area due to the relative simplicity to place these control volumes (CV) close to the surface and the log-law grow ratio that can be applied. This elements are

---

### Table 1: Mesh parameters.

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Mesh</th>
<th>NCV</th>
<th>$y_{\text{min}}^+$</th>
<th>$y_{\text{avg}}^+$</th>
<th>$y_{\text{min}}^+$</th>
<th>$y_{\text{avg}}^+$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Straight</td>
<td>Mesh 1</td>
<td>$2.13 \times 10^5$</td>
<td>2.98</td>
<td>111.97</td>
<td>9.24</td>
<td>106.28</td>
</tr>
<tr>
<td></td>
<td>Mesh 2</td>
<td>$1.11 \times 10^6$</td>
<td>0.13</td>
<td>18.44</td>
<td>2.27</td>
<td>18.66</td>
</tr>
<tr>
<td></td>
<td>Mesh 3</td>
<td>$7.83 \times 10^6$</td>
<td>0.04</td>
<td>8.51</td>
<td>0.34</td>
<td>9.63</td>
</tr>
<tr>
<td>Cone</td>
<td>Mesh 1</td>
<td>$2.20 \times 10^5$</td>
<td>2.37</td>
<td>98.52</td>
<td>12.31</td>
<td>98.89</td>
</tr>
<tr>
<td></td>
<td>Mesh 2</td>
<td>$1.10 \times 10^6$</td>
<td>0.18</td>
<td>15.50</td>
<td>1.56</td>
<td>13.99</td>
</tr>
<tr>
<td></td>
<td>Mesh 3</td>
<td>$8.00 \times 10^6$</td>
<td>0.04</td>
<td>6.90</td>
<td>0.48</td>
<td>7.33</td>
</tr>
</tbody>
</table>

$NCV$ the total number of control volumes in the domain, $y_{\text{min}}^+$ and $y_{\text{avg}}^+$ minimum and averaged $y^+$ on the valve disc, and $y_{\text{min}}^+$ and $y_{\text{avg}}^+$ minimum and averaged $y^+$ on the discharge orifice surface.
placed in the present case on the valve disc and discharge orifice geometry. Furthermore, the gap zone must be refined to correctly predict the flow discharge from the valve. As the flow moves away from the gap outlet the relative importance of the flow on the valve operation diminishes and so does the mesh resolution (see fig. 2). Three meshes are used with each valve model, see table 1 for details. Results in the present section have been obtained using the VMS turbulence model.

The fluid flows axially through the lower disk inlet orifice $d$ and thickness $h$, after that, impacts on the upper solid disk of diameter $D$ and finally, radially outflows through the gap between both coaxial disks. Present simulations use a turbulent inflow, which causes an unsteady turbulent behaviour characterized
Two important variables are shown in the numerical results presented. The first one is the fluid pressure distribution in the lower face of the upper disk, while the second one is the velocity magnitude distribution along the gap between coaxial disks at different radial positions of \( x = d/2 \) (gap position 1), \( x = d/4 + D/4 \) (gap position 2) and \( x = D/2 \) (gap position 3) corresponding to the entrance, middle and exit of the gap respectively.

Simulations are started from a solution obtained using a laminar inflow flow and are advanced in time until the initial transient behaviour is washed out at around \( 0.5TU \) \( (TU = tUref/d) \). Afterwards statistics have been collected and averaged by integrating the instantaneous data. Streamwise velocity time history and autocorrelation functions for the z-component of velocity (W) at the different locations are shown in figs. 4 and 5. It is interesting to observe the behaviour for the straight case, where probes P5 and P8, although being very close to each other show quite a different behaviour. P5 is located near the valve seat corner and exhibits large velocity fluctuations. P8, located slightly downwind is located at the beginning of the recirculation region and, although fluctuation magnitude is less, its frequency is considerably higher. P7 for the cone geometry is placed in a comparable location (just after the valve seat) and its turbulent behaviour is lower, i.e. lower fluctuation amplitude and frequency. This difference in flow behaviour is enhanced when observing probes P4 for the straight geometry and P9 for the cone geometry, both near the center of the gap. Flow for both geometries show some periodic like behaviour towards the valve outlet (see fig. 5a and 5b), however, as stated before, frequencies for the straight geometry seem higher than for the cone one.

Two important variables are shown in the numerical results presented. The first one is the fluid pressure distribution in the lower face of the upper disk, while the second one is the velocity magnitude (V) distribution along the gap between coaxial disks at different radial positions of \( x = d/2 \) (gap position 1), \( x = d/4 + D/4 \) (gap position 2) and \( x = D/2 \) (gap position 3) corresponding to the entrance, middle
Figure 6: Pressure distribution over the lower face of the valve disc and velocity profiles over the gap at $x = d/2, x = d/4 + D/4$ and $x = D/2$ for (a) straight geometry, (b) cone geometry.

and exit of the gap, all shown in fig. 6. Mesh independence is achieved for the $8 \times 10^6$ CV meshes as both pressure and velocity profiles show almost no change for a further mesh refinement. Stagnation pressure is significantly higher for the straight geometry than for the cone one, this is due to the higher velocity the flow reaches when passing through the straight valve. In the cone geometry this shape acts as a diffuser lowering the velocity of the impinging jet and easing into the direction change, thus, the flow carries less energy when crashing onto the top plate. Effects of these flow modifications are also visible through the gap. For the straight geometry flow reaches a higher velocity, additionally, the recirculation zone created by the valve seat corner is larger for the straight geometry than for the cone geometry (see figs. 6a and 6b).

4.2. Turbulence model analysis

As exposed in section 3.2, four turbulence models were used to evaluate the results in the flow through the valves, VMS, WALE, QR and SIGMA. Results show great consistency when changing LES models, however, this is more evident in the cone geometry than in the conical one. Sudden direction change and the more pronounced valve seat corner create a more complex flow that poses a higher difficulty to solve.

WALE and QR models predict a slightly larger pressure distribution than the rest of the models for the straight geometry, whereas VMS and WALE do the same in the cone geometry. For the velocity profiles results take a different trend. In this zone velocities for the straight geometry are larger than for the conical one due to the sudden change of direction and area reduction. For the conical geometry this acceleration is more gradual resulting in lower final velocities. All models results are quite similar for the entrance of the gap, however the other location analysed show results difference. SIGMA and VMS models show almost identical result in the remaining gap locations. For the straight geometry large differences are seen in gap positions 2 and 3. QR and WALE models both show larger velocities
in the recirculation region for both geometries. For the cone geometry these differences reduce for gap position 3, whereas for the straight geometry the differences become larger. VMS model shows results that fall in between the other models, much more evident for the velocity results than for the pressure ones. Previous experiences with turbulent flow suggest VMS results could be the best LES model to use for more detailed simulations.

4.3. Geometrical analysis
Changing the discharge orifice geometry greatly changes the velocity and pressure behaviour in the valve, as has been observed in the previous sections. To verify this behaviour different gap openings were studied as well, $s/d = 0.033$ and $s/d = 0.22$, using the same mesh size parameters than for

Figure 7: LES results for the pressure profile along the plate on (a) straight geometry, (b) cone geometry. LES results for the velocity profiles in the gap. (c) straight geometry, (d) cone geometry

Figure 8: pressure distribution over the plate for both geometries, (a): gap opening $s/d = 0.033$, (b): gap opening $s/d = 0.11$ and (c): gap opening $s/d = 0.022$
the base case. Gradual expansion allows the flow to change its direction sooner and reduce the energy carried when it impinges into the valve disc. This results in a considerably lower pressure rise in this element, as observed in fig. 8. This lower pressure rise over the valve disc for the cone geometry results in a considerably higher effective flow area (see fig.9), where for the straight geometry increases from 0.124 to 0.89 as the valve opens and for the cone geometry from 0.152 to 1.35. Effective force area also increases for both geometries, from 0.569 to 1.52 for the straight geometry and from 0.74 to 2.45 in the cone one.

Furthermore, the recirculation region in the valve seat is much smaller for the conical geometry. The sudden right angle change in the flow’s direction causes heavy flow separation. When the inlet channel geometry is changed to the cone the sudden direction change now possesses a smaller angle, thus, creating a smaller recirculation region. A second factor affecting this region is the velocity magnitude. As the flow is incompressible a reduction in flow area is accompanied by an increase in its velocity. Although, geometrically, the inlet and gap outlet areas are the same, effective flow areas are different due to the different velocity field and pressure drops present en each geometry (see fig.9). Observing fig. 10 there is a much larger velocity increment for the straight geometry than the conical one. As the flow opening increases from $s/d = 0.033$ to $0.11$ both effective flow and force areas increase as well, however, as the gap opening further opens to $s/d = 0.22$ this trend changes. Effective flow area for the cone geometry reduces its rate of increase, whereas the effective force area decreases. In the straight geometry, although both effective areas increases, its rate of increment diminishes.

Finally non-dimensional pressure drop ($C_P = \Delta P/0.5 \rho W_{ref}^2$) in the valve for the straight geometry and gap opening $s = 0.11d$ is $C_P = 4.11$, whereas for the cone geometry it is $C_P = 0.71$. This same behaviour remains when the valve opening is greater ($s/d = 0.22$) and smaller ($s/d = 0.033$), non-dimensional pressure drop for these valve openings and the straight geometry are $C_P = 1.26$ and $C_P = 65.32$ respectively, and for the cone geometry $C_P = 0.53$ and $C_P = 43.52$ respectively. Non-dimensional pressure drop reduces as the valve opens and is also significantly lower for the cone geometry, 57.9% for $s/d = 0.033$, 82.7% for $s = 0.11d$ and 76.4% for $s = 0.22d$.

5. Conclusions
Turbulent flow simulations through different valve geometries at an industrial type Re flow regime were carried out and their result compared to each other. Two mayor aspects were analysed and found to drastically change in shape and magnitude. Pressure profiles over the valve disc change from a nearly top-hat distribution to a smoother bell-shaped profile with the change of geometry. Additionally maximum pressure rise in this area is reduced nearly 40%. Velocity profiles in the gap show that for the straight geometry both, velocity magnitude and recirculation area, are larger then for the cone geometry. These effects greatly affect the valve operation, achieving a significant pressure drop reduction when using the cone geometry. Finally, it is important to note that the Mach number in some parts of the geometry surpass $Ma = 0.3$. In order to obtain better results compressible flow simulations are required.
Figure 10: Instantaneous velocity field for both geometries, (a): gap opening $s/d = 0.033$, (b): gap opening $s/d = 0.11$ and (c): gap opening $s/d = 0.022$

Acknowledgements
This work has been financially supported by the “Ministerio de Economía y Competitividad, Secretaría de Estado de Investigación, Desarrollo e Innovación”, Spain (reference ENE2010-17801), by the collaboration project between “Universitat Politècnica de Catalunya” and Termo Fluids S.L. and by the “Departamento Administrativo de Ciencia, Tecnología e Innovación - Colciencias” via their doctoral training program “Francisco Jose de Caldas”.

References


[34] Nicoud F, Toda H, Cabrit O, Bose S and Lee J 2011 Using singular values to build a subgrid-scale model for large eddy simulations *Phys Fluids* 23 085106