IX International Conference on Computational Heat and Mass Transfer, ICCHMT2016

The influence of the spatial discretization methods on the nozzle impulse flow simulation results

Piotr Cyklis*, Przemysław Mnlyarczyk

*Cracow University of Technology, Institute of Power and Process Engineering, Mechanical Faculty, al. Jana Pawla II 37, 31-864 Krakow, Poland

Abstract

Vibration and noise caused by pressure pulsations in the volumetric compressor manifolds are one of the most important problems in the volumetric compressor’s operation. For pressure pulsation attenuation, different types of mufflers are applied using a design based on the Helmholtz resonator approach. This design is not effective for variable revolution speed compressors and in this case other pressure pulsations attenuation methods are required. Nowadays, due to the increasing application of variable speed compressors new design methods are developed. One of the possibilities to attenuate pressure pulsations over a wide range of frequencies is the introduction of specially shaped nozzles in the gas duct flow directly after the compressor outlet chamber. It is well known that a nozzle can attenuate pressure pulsations, however, it increases the compressor power at the same time. The main criterion for the nozzle selection is the highest possible attenuation of pressure pulsations coupled with the lowest possible effect on the compressor pumping power. In some cases 25-30% of the pressure damping is good enough to reach the standard requirements. The method applying CFD impulse flow simulations has been developed for the estimation of the nozzle influence on pressure pulsations for all frequencies. The FLUENT/ANSYS software has been used for the simulation. The CFD simulation requires a lot of effort for proper selection of the boundary conditions, solution methods, turbulence models, discretization methods, mesh design etc. The results are in some cases surprisingly different between different methods. In this paper the flow simulations for the nozzle flow with the connecting pipe using impulse function as a mass inflow excitation is shown. In this investigation project various models and methods have been compared and results in 2D and recalculated into 0D time only dependent functions are presented.

© 2016 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).

Keywords: CFD simulations, Nozzle flow, Numerical methods, pressure pulsations

Nomenclature

\[ \hat{A} \] surface area vector

Corresponding author: pcyklis@mech.pk.edu.pl
1. Introduction

The CFD simulation is a powerful tool which can be used for different purposes. In most cases the aim is to look deeply into the process inside a device or a machine. In the present case, however, the aim of the simulation is to assess the influence of the manifold element on pressure pulsation attenuation or amplification (which is also possible). Therefore the requirements for the CFD method are different. It is more important to see how the inlet impulse treated as excitance is transmitted to the outlet response. Periodic work flow of volumetric compressors causes the mass flow pulsations. The pulsating flow means also pulsating pressure which is the source of vibration and noise, which is one of the most important problems in compressor operation nowadays [1][2][3]. In the paper [4] a CFD simulation of a single pipe excited with a single disturbance is shown. The response, which is periodic with a constant frequency, is characterized by a certain degree of damping. The paper shows that the analysis of pressure pulsation damping by the different elements is important.

The response signal obtained in the simulation of impulse flow in a nozzle can help to determine the effect of the nozzle shape on pressure pulsations damping. The main aim of the present study is to demonstrate how the selection of the numerical method affects the results. In particular, the impact of spatial discretization on the system response is presented.

2. CFD Methods

FLUENT/ANSYS provides comprehensive modeling capabilities for a wide range of incompressible and compressible, laminar and turbulent fluid flow problems. The choice for specific codes for the fluid flow, turbulence model for the unsteady flow are possible [5]. There are suggestions for the user for most popular cases, however, for special applications the user has to choose on the basis of their own expertise.

For the impulse flow the conservation equations for mass (1), momentum (2)(3) and energy (4) are solved. The general form of the mass conservation equation for 2D axisymmetric geometries, which is valid for both compressible and incompressible flow, can be written as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho v_x) + \frac{\partial}{\partial r}(\rho v_r) + \frac{\rho v_r}{r} = S_m$$  \hspace{1cm} (1)

where the source $S_m$ is the mass added to the continuous phase from the dispersed second phase and any user-defined sources, but in the present case this value is zero. The radial and axial momentum conservation equations in an inertial reference frame are described by:

$$\frac{\partial}{\partial t}(\rho v_x) + \frac{1}{r} \frac{\partial}{\partial x}(r \rho v_x v_x) + \frac{1}{r} \frac{\partial}{\partial r}(r \rho v_r v_x) = -\frac{\partial p}{\partial x} + \frac{1}{r} \frac{\partial}{\partial r}\left[r \mu \left(2 \frac{\partial v_x}{\partial x} - \frac{2}{3} (\nabla \cdot \nu)\right)\right] + \frac{1}{r} \frac{\partial}{\partial r}\left[r \mu \left(\frac{\partial v_x}{\partial r} + \frac{\partial v_r}{\partial x}\right)\right] + F_x$$  \hspace{1cm} (2)
The energy equation is solved in the form:
\[
\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\mathbf{v}(\rho E + p)) = \nabla \cdot \left( k_{\text{eff}} \nabla T + (\tau_{\text{eff}} \cdot \mathbf{v}) \right) \tag{4}
\]

Two terms on the right-hand side of equation (4) represent energy transfer due to conduction and viscous dissipation, respectively. \( E \) is defined by equation (5):
\[
E = h - \frac{p}{\rho} + \frac{v^2}{2} \tag{5}
\]

In [6] the impulse flow with different turbulence models was compared. It was found that for the impulse flow simulated with different turbulence models the differences are negligible. For the inviscid flow simulations the amplitudes are a little higher, however, this does not affect the comparison of qualitative results. As the calculations take the same amount of time for different turbulence models, to calculate the impulse flow for the expected geometries and boundary conditions the Raynolds Stress model, as the most efficient, was selected for the studies presented here.

The RSM turbulence model for 2D flows is based on five additional transport equations which close the Reynolds-averaged Navier-Stokes equations.

2.1. The Solver

For the impulse flow simulation the FLUENT/ANSYS density-based solver was used. To meet the convergence criteria each iteration consists of five steps [5]. At the beginning the solver updates the fluid properties basing on the solution initialization (for the first step) or on the previous step. Next, the energy, momentum and continuity equations are solved. Using RSM turbulence model the solver has to solve equations for turbulence, and then the convergence of the equation set is checked. This scheme is continued until the convergence criteria are fulfilled. In the presented simulations the explicit formulation was used. The main difference between the density-based explicit and implicit approaches is that in the explicit formulation all the variables are solved in one cell at a time whereas in using the implicit formulation all the variables are solved in all cells.

2.2. Spatial discretization methods

For the scalar quantity \( \phi \) the discretization of governing equations in an integral form for an arbitrary control volume \( V \) can be presented as [5]:
\[
\int_V \frac{\partial \rho \phi}{\partial t} dV + \oint_{\Gamma_v} \rho \phi \mathbf{v} \cdot d\mathbf{A} = \oint_{\Gamma_s} \nabla \phi \cdot d\mathbf{A} + \int_V S_{\phi} dV \tag{6}
\]

Discretization of this equation for general two-dimensional cell is presented in the equation (7):
\[
\frac{\partial \rho \phi}{\partial t} V + \sum_{f}^{N_f,\text{faces}} \rho_f \mathbf{v}_f \cdot \Gamma_f \mathbf{A}_f = \sum_{f}^{N_f,\text{faces}} \Gamma_f \nabla \phi \cdot \mathbf{A}_f + S_{\phi} V \tag{7}
\]
The FLUENT/ANSYS stores discrete values of the scalar \( \varphi \) at the cell centers so the face values \( \varphi_f \) are interpolated from the cell center values as this is the program default solution. This is achieved by the application of one of the available spatial discretization schemes.

To compute the gradient of a given variable three different methods are included in the FLUENT/ANSYS: The Green-Gauss Cell-Based, The Green-Gauss Node-Based and the Least Squares Cell-Based.

In the presented investigation the program default method, which is Least Squares Cell-Based, has been selected. The accuracy of this method is comparable with the others but in same cases it is less expensive in terms of the simulation time. The main idea of this method is that the solution varies linearly. The change of the values between the two cells (as shown in figure 1.) along the known vector is expressed by equation (8):

\[
(V\Phi)_{c0} \cdot \Delta r_i = (\Phi_{cI} - \Phi_{c0})
\]  

(8)

For the computation of different variables like Turbulent Kinetic Energy, Turbulent Dissipation Rate or Raynolds Stresses different spatial discretization schemes are available:

- First-Order Upwind Scheme – which is a default scheme in ANSYS/Fluent and was chosen as the basic scheme of the impulse flow simulations. In the presented case the results obtained using First-Order Upwind and Second-Order Upwind scheme was similar, so the First-Order Upwind scheme was chosen as less time-consuming. This scheme is based on the principle that the face quantities are identical to the cell quantities. It means that the cell-center values of any field variable represent a cell-average value and hold throughout the entire cell.

- Second-Order Upwind Scheme,
- Power-Law Scheme,
- Third-Order MUSCL,
- QUICK Scheme – which is the scheme dedicated to the quadrilateral and hexahedral meshes, where the subsequent cells can be specified. The scheme is based on a weighted average of second-order-upwind and central interpolations of the variable. It can be written, for the face e from figure 2, as :

\[
\Phi_e = \Theta \left[ \frac{S_p}{S_p + S_e} \Phi_p + \frac{S_e}{S_p + S_e} \Phi_e \right] + (1 - \Theta) \left[ \frac{S_p + 2S_e}{S_p + S_e} \Phi_p - \frac{S_p}{S_p + S_e} \Phi_W \right]
\]  

(9)

and the letters in the subscript describe which quantity is related to a given value, as it is in the figure 1.

In FLUENT/ANSYS the value of \( \Theta \) depends on the solution and for the traditional QUICK scheme is 0.125. The QUICK, Power Law and MUSCL schemes give similar results in the presented case, therefore the QUICK scheme was selected as an alternative scheme to conduct the impulse flow simulations. The comparison of results obtained for the simulations using QUICK and First-Order Upwind schemes is the main problem presented in this publication.

For transient simulations, the governing equations are discretized in both time and space. The explicit time integration for the density-based solver is described by:

\[
\frac{\Phi^{n+1} - \Phi^n}{\Delta t} = F(\Phi^n)
\]  

(10)
where $F$ incorporates any spatial discretization. Such an approach is preferred to capture the behavior of moving waves [5].

3. Numerical model

All of the investigated simulations were conducted in FLUENT/ANSYS software. For all cases the 2D axisymmetric, ideal gas isentropic flow model was used with the Reynolds-Stress turbulence model. To avoid the introduction of too many variables, for the RSM transport equations the software default discretization models was selected. The simulation was transient with a time step equal to $2\times10^{-6}$ second, and the total simulation time was 0.01 [s]. The impulse flow damping using different spatial discretization schemes was conducted for different damping elements. The investigated nozzle shapes with the inner diameter of 20 mm are the Venturi orifice, Venturi nozzle and hyperboloidal nozzle. The outer diameter of the nozzles is 35 mm and the installation pipe has the same diameter. The main goal of these investigations is to find the best numerical method to simulate the nozzle impulse flow damping. As a verification of the simulations results the experimental investigations were conducted for the same nozzles shapes.

3.1. Mesh

The appropriate mesh was prepared for every geometry. As it is 2D axisymmetric simulations, the model contains the nozzle shape and straight pipe fragment, according to the test stand installation. In figure 2 the simulated elements of the mesh are presented.

![Fig. 2. Different mesh types for shapes: a) Venturi nozzle, b) Venturi orifice, c) Hyperboloidal nozzle](image)

The geometrical model and mesh were the same for each shape for both simulations. In figure 3 different meshes are presented in order to clarify that each of these meshes are correct in this case, and changing grid size, adding the near wall layers etc. does not affect the result of the impulse flow simulation. The most important issue was to obtain structured mesh, according to the QUICK discretization scheme requirements. Authors checked the mesh for various combinations of grid size and for different grid types in this simulations. For the presented geometries, good quality mesh was obtained for approximately 3500 cells with the cell size $\sim1$ mm.

3.2. Boundary conditions

The numerical model has to be simple and fast to compute, yet accurate for the purpose. The model is 2D and simple, therefore there are only four necessary boundary conditions:

- Mass flow Inlet – where the mass flow impulse excitation is $0.1$ [kg/s]. The impulse excitation means that its duration is equal to the first, one time step. For the rest of these transient simulation steps the mass flow inlet boundary is defined as zero. The backflow temperature, if it does occur is defined as 350 K – close to the flow temperature in the experimental test stand.

- Pressure outlet – where the pressure at the outlet is defined as the arithmetical average between pressure outside the domain and the last cell inside the domain. The backflow temperature, if it does occur is defined as 350K.

- Wall – where the tangential stresses are included in the momentum conservation equation,
- **Axis** – to determine the appropriate physical value for a particular variable at a point on the axis, the software uses the cell value in the adjacent cell.

### 4. Numerical results

For the presented cases numerical simulations were conducted. For each element the results were spatially averaged at the outlet to obtain one-dimensional mass flow fluctuations which are the result of the impulse flow excitation. The damping coefficients were estimated for each free frequency, using the method described in [7].

The simulations were conducted for two different spatial discretization methods, first, using the First-Order Upwind scheme, which will be called “basic”, for which the obtained signals with attenuation curves are shown in figure 3, and then for the QUICK scheme, for which the results are shown in figure 4.

![Fig. 3. Attenuation curves for basic simulations. Venturi orifice (left), Venturi nozzle (center), hyperboloidal nozzle (right).](image1)

As can be seen, the attenuation curves for basic simulations are not similar, as the charts are unbalanced. The damping factors for the upper and lower attenuation curves differ significantly. This provokes a question which attenuation curve better describes the damping in this case. Or maybe it is better to calculate the average value of the damping factor for two curves, and thus the question arises whether geometric or arithmetic average value? The results are not clear. Comparing the simulation results, where the QUICK discretization scheme was used, it looks much easier to analyze.

![Fig. 4. Attenuation curves for QUICK scheme simulations. Venturi orifice (left), Venturi nozzle (center), hyperboloidal nozzle (right).](image2)

In the simulation results where QUICK scheme was used, the attenuation curves are similar, hence the damping coefficients are similar. In this case, the ordinary averaging of their values can be considered as a reliable attenuation value.

The results for the damping coefficient obtained in the basic simulations are summarized in table 1.

<table>
<thead>
<tr>
<th>Damping factor</th>
<th>Venturi orifice</th>
<th>Venturi nozzle</th>
<th>Hyperboloidal nozzle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Upper curve</td>
<td>-0.08</td>
<td>-0.031</td>
<td>-0.055</td>
</tr>
</tbody>
</table>
There is a significant difference between the damping factor value for the upper and lower attenuation curves. However, the trend for both averaged values is similar, i.e. the damping factor is the highest for the hyperboloidal nozzle and smallest for the Venturi nozzle. The results of the simulations using QUICK scheme are summarized in Table 2.

<table>
<thead>
<tr>
<th>Damping factor</th>
<th>Venturi orifice</th>
<th>Venturi nozzle</th>
<th>Hyperboloidal nozzle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Upper curve</td>
<td>-0.051</td>
<td>-0.027</td>
<td>-0.034</td>
</tr>
<tr>
<td>Lower curve</td>
<td>-0.05</td>
<td>-0.45</td>
<td>-0.026</td>
</tr>
<tr>
<td>Arithmetic average</td>
<td>-0.0505</td>
<td>-0.036</td>
<td>-0.03</td>
</tr>
<tr>
<td>Geometric mean</td>
<td>0.05</td>
<td>0.03</td>
<td>0.03</td>
</tr>
</tbody>
</table>

As shown in Table 2, the results are more structured. In Figure 5 the comparison between the results of these two simulations are presented.

![Figure 5](source_url)

In the presented figures, only for the upper attenuation curve the damping factor varies in a similar manner for different shapes in both numerical investigations. For all the other values the obtained results, for different discretization schemes, are not similar in any way. To find out the most reliable simulation method, comparisons with the results of experimental investigation are necessary.

5. Experimental verification

The experimental investigations were performed on a special test stand prepared to measure, in particular, pressure pulsations in the compressor discharge manifold. The test stand and the tests themselves were described in detail in publications [7][8]. Due to the limited space here only the final results of the experimental tests are presented. The effect of the nozzle on the pressure pulsations was measured on the test stand for different compressor revolution speeds. In this paper the results for only one revolution speed are shown for comparison. The test stand consists of a DEMAG screw compressor with specially prepared discharge installation, with a place for nozzle mounting. On the test stand all the necessary measuring devices (like dynamic pressure sensors, metering orifices etc.) were mounted. The results of the pressure pulsations damping for the compressor revolution speed of
1615 rev/min are presented. The pressure pulsations values in the installation with different nozzles are presented in figure 6. The figure 6 also show the comparison of the experimental investigation with the results obtained using the simulation. The values are the percentage of the damping gain referring to the hyperboloidal nozzle damping – as it shows the least attenuation behavior in the experimental investigations.

The QUICK method for the presented problem gives results comparable with the experimental results, however, there are still quantitative differences. This is due to the fact that simulation is computed for impulse excitation, which means that it corresponds with all the frequencies, and the experimental results are for one frequency only. Nevertheless, the QUICK method gives the damping assessment for the nozzle for the compressor manifold which can be used for optimization. The impulse excitation gives the possibility to assess the variable rotation speed of the compressor.

6. Conclusions

The presented results of the numerical simulations show that for the impulse flow simulations spatial discretization schemes have a significant impact on the results. The conclusion from our computations was that the use of different turbulence models, changing time step or changing the mesh are negligible for the impulse flow simulation. In this investigation the results of two different spatial discretization methods were presented: the First-Order Upwind scheme called “basic” and for the QUICK scheme. It was found out that the results from the “basic” method are completely unacceptable for the impulse flow simulation, while the QUICK scheme gives results which can be compared with the experimental results. This method can be used to optimize the shape and dimensions of the manifold element for pressure pulsations damping.

7. References