Modelling of Transonic Flow in Very Narrow Channels

Jan Vimmr

Department of Mechanics, University of West Bohemia in Pilsen, Univerzitní 22, CZ-306 14 Plzeň, Czech Republic

The study is focused on the mathematical modelling of leakage flow in very narrow channels. The problem is solved as a two-dimensional non-stationary turbulent compressible Newtonian fluid flow with ideal gas properties. The turbulent flow is assumed to be statistically steady and the mathematical model is described by the non-linear conservative system of the compressible Favre-averaged Navier-Stokes equations. Its numerical solution is performed using the explicit cell-centred finite volume time stepping scheme of MacCormack algorithm on a structured quadrilateral grid. For the computation of turbulent viscosity, the algebraic Baldwin-Lomax turbulence model is implemented into the presented numerical code. The leakage flow is studied for two given pressure ratios between inlet and outlet of a computational domain.

Keywords: clearance gaps, gas leakage, compressibility, Favre-averaged Navier-Stokes equations, algebraic turbulence model, finite volume method, explicit MacCormack scheme

Introduction

Mathematical modelling of transonic flow of viscous compressible fluids in very narrow channels is one of the very topical and demanding problems of internal aerodynamics today. Screw compressors, Fig. 1 (left), represent one of many applications. The most important part of screw compressors is its work space. It has a complex geometry [8], which volume decreases during the rotors motion. That causes the compression of the fluid. The work chambers are separated by three main types of clearance gaps – frontal gaps at axial ends of rotors, gaps between rotors themselves and gaps between rotors and housing. The processes, which take place in work chambers and especially in clearance gaps on their boundaries, have a significant influence on the screw compressor performance. The knowledge of details of the leakage flow through clearance gaps is essential to make reasonable estimates for mass flow rate and to define the loss of medium. Therefore, the numerical and experimental investigation of the complex clearance flow is necessary from a practical point of view. The clearance flow can be reasonably modelled by computational fluid dynamics for so called dry compressors, Fig. 1 (left), where no multiphase flow occurs.

The leakage flow through clearance gaps is studied by many screw compressor engineers. In [6] and [7] some experimental and numerical simulations of compressible viscous fluid flow through a 2-D model of the male rotor-housing gap in a screw compressor for various pressure ratios are presented. The male rotor-housing gap is the sealing gap between the head of the male rotor tooth and housing shown in a screw compressor frontal cross-section as a number 5 in Fig. 1 (right). All of these...
computations were only performed by available commercial computational system Fluent using various turbulence models and the rotary motion of the male rotor was not involved in the computations.

In [10], the author has already presented the laminar computation of the leakage flow through the 2-D model of the male rotor-housing gap, Fig. 2, using the own developed numerical code. The width of this gap in its narrowest section equals to 100 µm. The laminar computation was performed for the reference Reynolds number \( \text{Re}_{\text{ref}} = 32,941 \) and for the given pressure ratio \( p_1 / p_2 = 2 \) with atmospheric pressure at the outlet. The obtained results shown that the assumption of the laminar leakage flow computation in this 2-D model of the clearance gap is not exactly correct.

In the present paper, therefore, based on the previous results by the author, [10], research attention is focused on the modelling of a turbulent compressible viscous fluid flow through the 2-D model of the male rotor-housing gap, Fig. 2, for two various pressure ratios \( p_1 / p_2 = 2 \) and \( p_1 / p_2 = 3 \) with atmospheric pressure at the outlet. Because of the absence of experimental data the computed numerical results are compared with the results obtained by the commercial CFD package Fluent.

![Dry screw compressor (left) and screw compressor frontal cross-section (right)](image)

![Two-dimensional computational domain \( \Omega \subset R^2 \) with a structured quadrilateral grid (170x68 cells)](image)

<table>
<thead>
<tr>
<th>Nomenclature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( a )</td>
<td>speed of sound</td>
</tr>
<tr>
<td>( A, B )</td>
<td>Jacobian matrices of inviscid fluxes ( f, g )</td>
</tr>
<tr>
<td>( c_p )</td>
<td>specific heat at constant pressure</td>
</tr>
<tr>
<td>( c_v )</td>
<td>specific heat at constant volume</td>
</tr>
<tr>
<td>( E )</td>
<td>total energy per unit volume</td>
</tr>
<tr>
<td>( F^i, k = 1,2 )</td>
<td>components of inviscid flux ( F^i )</td>
</tr>
<tr>
<td>( F^v, k = 1,2 )</td>
<td>components of viscous flux ( F^v )</td>
</tr>
<tr>
<td>( f, g )</td>
<td>Cartesian components of inviscid flux ( F^i )</td>
</tr>
<tr>
<td>( f_v, g_v )</td>
<td>Cartesian components of viscous flux ( F^v )</td>
</tr>
<tr>
<td>( k )</td>
<td>thermal conductivity</td>
</tr>
<tr>
<td>( L_{\text{ref}} )</td>
<td>Characteristic length</td>
</tr>
<tr>
<td>( M )</td>
<td>Mach number</td>
</tr>
<tr>
<td>( u, v )</td>
<td>Cartesian components of velocity vector</td>
</tr>
<tr>
<td>( w )</td>
<td>vector of conservative variables</td>
</tr>
<tr>
<td>( w_n^* )</td>
<td>numerical solution in the centre of the cell ( \Omega ) at time ( n )</td>
</tr>
<tr>
<td>( y = (y_1, y_2)^T )</td>
<td>vector of space coordinates</td>
</tr>
<tr>
<td>( \alpha )</td>
<td>angle of attack of the flow</td>
</tr>
<tr>
<td>( \delta_{\text{K}} )</td>
<td>Kronecker delta</td>
</tr>
<tr>
<td>( \phi )</td>
<td>flow variable</td>
</tr>
<tr>
<td>( \Gamma, m = 1,\ldots,4 )</td>
<td>edges of the quadrilateral cell ( \Omega )</td>
</tr>
<tr>
<td>( \eta )</td>
<td>molecular viscosity</td>
</tr>
<tr>
<td>( \eta_t )</td>
<td>turbulent viscosity</td>
</tr>
<tr>
<td>( \kappa )</td>
<td>Poisson’s constant</td>
</tr>
</tbody>
</table>
\[ n = (n_x, n_y) \] outward unit normal vector
\[ \rho \] density
\[ \Omega \subset \mathbb{R}^2 \] 2-D computational domain
\[ \Omega_t \] space time cylinder
\[ \Omega_y \] quadrilateral cell (finite volume)
\[ |\Omega_y| \] face area of the quadrilateral cell \( \Omega_y \)
\[ \partial\Omega \] boundary of the computational domain
\[ \tau_{n} \] shear stresses

Subscripts
1 inlet
2 outlet
\( i, j \) spatial indexes
\( W \) denotes adiabatic impermeable wall
\( \text{ref} \) reference quantities

Problem Formulation

The male rotor-housing gap is modeled in screw compressor frontal cross-section (the plane perpendicular to rotors axes), Fig. 1 (right), by a two-dimensional computational domain \( \Omega \subset \mathbb{R}^2 \), Fig. 2. It is bounded by two adiabatic impermeable walls \( \partial\Omega_{ws} = \partial\Omega_{ws} \cup \partial\Omega_{ws} \) that corresponds to the circular part of the housing \( \partial\Omega_{ws} \) and to the head of the male rotor tooth \( \partial\Omega_{ws} \). The other boundaries represent the inlet \( \partial\Omega_i \) from the higher pressure chamber and outlet \( \partial\Omega_o \) to the lower pressure chamber. The width of this 2-D channel in its narrowest section equals to 100 \( \mu \)m. The turbulent compressible viscous fluid flow through the 2-D computational domain \( \Omega \subset \mathbb{R}^2 \), Fig. 2, is studied for two given pressure ratios \( p_{in}/P = 2 \) and \( p_{in}/P = 3 \) with atmospheric pressure at the outlet.

Mathematical Model

Let \( \Omega \subset \mathbb{R}^2 \) be a 2-D computational domain with the boundary \( \partial\Omega = \partial\Omega_i \cup \partial\Omega_o \cup \partial\Omega_y \) and \( (0,T) \) be a time interval. Then \( \Omega_t = \Omega \times (0,T) \) denotes the space time cylinder. In the laminar case, the motion of a compressible, viscous, heat-conducting, Newtonian fluid in an absolute frame of reference is described by the non-linear conservative system of the Navier-Stokes (NS) equations derived from the integral form of the conservation laws for mass, momentum and total energy in Eulerian description.

In order to obtain the governing equations for turbulent compressible flow, it is convenient to replace the instantaneous quantities in the conservative system of the NS equations by their mean and their fluctuating values. In conventional time averaging introduced by Reynolds, the instantaneous flow variable \( \Phi(y,t) \) is expressed as the sum of a mean \( \bar{\Phi}(y,t) \) and a fluctuating part \( \Phi'(y,t) \), so that

\[ \Phi(y,t) = \bar{\Phi}(y,t) + \Phi'(y,t), \quad (1) \]

\[ \bar{\Phi}(y,t) = \frac{1}{\Delta t} \int_{t}^{t+\Delta t} \Phi(y,t) dt. \quad (2) \]

In mass-weighted time averaging suggested by Favre, the instantaneous flow variable \( \Phi(y,t) \) is decomposed into the mass-averaged part \( \bar{\Phi}(y,t) \) and a fluctuating part \( \Phi'(y,t) \), wherefore

\[ \Phi(y,t) = \bar{\Phi}(y,t) + \Phi'(y,t), \quad (3) \]

\[ \bar{\Phi}(y,t) = \frac{\rho \bar{\Phi}}{\rho}. \quad (4) \]

Favre-Averaged Navier-Stokes Equations

Applying the conventional time average decomposition Eq.(1) of density \( \rho \) and static pressure \( p \) and the mass-weighted time average decomposition Eq.(3) of the velocity vector \( \Phi \), total energy per unit volume \( E \) and thermodynamic temperature \( T \) together with application of the averaging operations described in [11] or [4], produces the non-linear system of the Favre-averaged Navier-Stokes (FANS) equations written in non-dimensional conservative form as

\[ \frac{\partial w}{\partial t} + \sum_{i=1}^{3} \frac{\partial F'_i(w)}{\partial y_i} = \frac{1}{\text{Re}_{ref}} \sum_{i=1}^{3} \frac{\partial F'_i(w)}{\partial y_i} \text{ in } \Omega_t. \quad (5) \]
In order to close the non-linear system of the governing equations, it is usually assumed constant and for wall bounded flows is \( \text{Pr}_t = 0.9 \).

Assuming a calorically perfect gas, the static pressure is given by the equation of state
\[
\rho = \rho(rT) \equiv (k-1) \left( \frac{\tilde{E}}{\rho} - \frac{1}{2} \rho \tilde{v}^2 \right),
\]
where \( k = 1.4 \) and the laminar Prandtl number defined as \( \text{Pr}_t = c_p \eta/k = 0.72 \) is taken to be a constant for a calorically perfect gas. The external volume forces and molecular viscosity fluctuations are not considered in our application.

It remains to determine the turbulent viscosity \( \eta_t \). The complexity of our problem is a motivation for the use of relatively simple turbulence models. The advantage of algebraic models is that no additional transport differential equations have to be solved. In this study, the algebraic turbulence model introduced by Baldwin and Lomax, [1], is considered. It is a two-layer turbulence model, based on the mixing-length hypothesis, which is formulated for use in computations where boundary layer properties are difficult to determine.

### Numerical Method

The computational domain \( \Omega \subset R^2 \) is divided into a finite number of small, non-overlapping, quadrilateral cells \( \Omega_i \). Let us consider a partition \( 0 < t_n < t_{n+1} < \ldots < t_{n+4} \) of the time interval \( (0,T) \) and set \( \Delta t = t_{n+1} - t_n \). For the discretization of the non-linear conservative system of the FANS equations Eq.(5) the cell-centred finite volume method on a structured quadrilateral grid, Fig. 2, is used. Time integration of the inviscid part of the system Eq.(5) is realized by the finite volume formulation of the frequently used implicit two-step MacCormack scheme. The approximation of the viscous part of the system Eq.(5) is modeled using a finite volume version of central differences on dual cells, see [9] for details.

\[
\text{Visc} (\omega_i) = \frac{1}{Re_{ref}} \sum_{j=1}^4 \left( f_{ij} S^+_u + g_{ij} S^-_u \right)
\]

and is added to the predictor and corrector steps of the MacCormack algorithm

\[
\omega_i^{n+1} = \omega_i^n - \frac{\Delta t}{|\Omega_i|} \sum_{j=1}^4 \left( f_{ij} S^+_u + g_{ij} S^-_u \right) + \\
\frac{\Delta t}{|\Omega_i|} \text{Visc} (\omega_i^n)
\]

\[
\omega_i^{n+1} = \omega_i^n + \frac{1}{2} \left[ \omega_i^n + \omega_i^{n+1} \right] - \frac{\Delta t}{|\Omega_i|} \sum_{j=1}^4 \left( f_{ij}^{n+1} S^+_u + g_{ij}^{n+1} S^-_u \right) + \\
\frac{1}{2} \frac{\Delta t}{|\Omega_i|} \text{Visc} (\omega_i^{n+1})
\]

where \( \omega_i^{n+1} \) is the corrected numerical solution at time \( t_{n+1} \). In our application we use the dissipative terms proposed by Causon, [2]. The Cartesian components \( f_u \) and \( g_v \) of the inviscid numerical fluxes \( F_{ij}^v \) through the edges \( \Gamma_{ij}^v \), \( m = 1, \ldots, 4 \) of the cell \( \Omega_i \) at time \( t_n \) are evaluated as

\[
f_i^v = f (w_{i,n}^u), \quad f_2^v = f (w_{i+1,n}^u), \quad f_3^v = f (w_{i,n}^u), \quad g_i^v = g (w_{i,n+1}^u), \quad g_2^v = g (w_{i+1,n+1}^u), \quad g_3^v = g (w_{i,n+1}^u),
\]

and at time \( t_{n+1} \) as

\[
f_i^{n+1} = f_i^{n+1} = f (w_{i,n+1}^u), \quad f_2^{n+1} = f (w_{i+1,n+1}^u), \quad f_3^{n+1} = f (w_{i,n+1}^u), \quad g_i^{n+1} = g (w_{i,n+1}^u), \quad g_2^{n+1} = g (w_{i+1,n+1}^u), \quad g_3^{n+1} = g (w_{i,n+1}^u),
\]
was chosen in the stability condition and \( \Delta y_i = 0.16 \sqrt{Re} \). It can be seen that the leakage flow is \( n_{ij} = 0 \) in the vicinity of the solid wall have the width given by the formula \( \Delta y_{ij} = (1 + A) \Delta y_i \), where \( A = 0.17 \), \( Re = 40000 \) and \( s = 1, \ldots, 28 \).

The problem of leakage flow through the computational domain \( \Omega \subset R^2 \), Fig. 2, was solved for two given pressure ratios \( \bar{p}_{in}/\bar{p}_z = 2 \) and \( \bar{p}_{in}/\bar{p}_z = 3 \) as non-stationary turbulent compressible Newtonian fluid flow. The following non-dimensional boundary conditions were prescribed – at the inlet \( \partial \Omega_i \) : the total pressure \( \bar{p}_{in} = 1 \), the total temperature \( \bar{T}_{in} = 1 \), the inlet angle \( \alpha \) of attack of the flow, \( \partial \bar{T}/\partial n = 0 \) and \( \sum_{i=1}^3 \bar{z}_i n_i = 0 \), \( g = 1.2 \); at the outlet \( \partial \Omega_o \) : the static pressure \( \bar{p}_o = 0.5 \) (in the first case) and \( \bar{p}_o = 0.333 \) (in the second case), \( \partial \bar{T}/\partial n = 0 \) and \( \sum_{i=1}^3 \bar{z}_i n_i = 0 \), \( g = 1.2 \); on the solid walls \( \partial \Omega_{ws} \) and \( \partial \Omega_{so} \) : \( \bar{u} = 0 \), \( \bar{v} = 0 \) and \( \partial \bar{T}/\partial n = 0 \). \( n \) is the outward unit normal vector to the boundary.

The free stream conditions were imposed over the entire computational domain \( \Omega \subset R^2 \) in order to initialize the explicit two-step TVD MacCormack scheme, Eqs.(13) – (15), proposed by Causon, [2]. The turbulent viscosity \( \eta \) was computed using the algebraic Baldwin-Lomax turbulence model which is mathematically simple and its implementation is easy. In our application, the typical model constants, [1], were used. The parameter \( CFL = 0.1 \) was chosen in the stability condition Eq.(17).

The numerical results obtained for the pressure ratio \( \bar{p}_{in}/\bar{p}_z = 2 \) are presented in Fig. 3 – Fig. 7. In this case, the reference Reynolds number \( Re_{av} = 32941 \) was computed from the selected reference values \( \rho_{av} = 202650 \text{ Pa} \), \( T_{av} = 373.15 \text{ K} \), \( L_{av} = 0.001 \text{ m} \), \( r_{av} = 287 \text{ J kg}^{-1} \text{ K}^{-1} \) and \( \eta_{av} = 1.879 \cdot 10^{-5} \text{ kg m}^{-1} \text{ s}^{-1} \), which gives \( \rho_{av} = \rho_{av} / (r_{av} T_{av}) = 1.8923 \text{ kg m}^{-3} \) and \( u_{av} = \sqrt{\rho_{av} / \rho_{av}} = 327.25 \text{ m s}^{-1} \). Fig. 3 displays the isolines of the Mach number distribution computed in the 2-D model of the male rotor-housing gap and plotted with \( \Delta M = 0.03 \). It can be seen that the leakage flow is transonic (\( M_{av} = 1.2 \)). The velocity magnitude distribution in the 2-D computational domain \( \Omega \) is shown in Fig. 4. The static pressure distribution along the center-
line of the clearance gap and the isolines of the static pressure computed in the whole computational domain \( \Omega \) and plotted with \( \Delta p = 3500 \, \text{Pa} \) are shown in Fig. 5. The static pressure and Mach number distributions along the centerline of the male rotor-housing gap computed by the presented explicit TVD MacCormack scheme with algebraic Baldwin-Lomax turbulence model are compared in Fig. 6 and Fig. 7 with results obtained on the same computational grid in commercial CFD package Fluent using Spalart-Allmaras and RNG \( k-\varepsilon \) turbulence models. Resulting pressure distribution obtained by the Baldwin-Lomax model, Fig. 6, shows an unphysical sharp change of pressure at the end of the gap. That may be a reason that the pressure minimum is overestimated while the maximal velocity is underestimated using this model, Fig. 7.

The numerical results gained for the pressure ratio \( \overline{p}_1/\overline{p}_2 = 3 \) are presented in Fig. 8 – Fig. 10. For this case, the reference Reynolds number \( Re_{ref} = 49416 \) was computed from the selected reference values \( p_{ref} = 503.975 \, \text{Pa} \), \( T_{ref} = 373.15 \, \text{K} \), \( L_{ref} = 0.001 \, \text{m} \), \( r_{ref} = 287 \, \text{J/kg}^\circ \text{K} \) and \( \eta_{ref} = 1.879 \times 10^{-3} \, \text{kg/m}^2\text{s} \), which gives \( p_{ref} = p_{ref}/(r_{ref} T_{ref}) = 2.8384 \, \text{kg/m}^3 \) and \( u_{ref} = \sqrt{p_{ref}/\rho_{ref}} = 327.25 \, \text{m/s} \). Fig. 8 displays the isolines of the Mach number distribution computed in the 2-D model of the clearance gap and plotted with \( \Delta M = 0.03 \). The maximum value of the Mach number is \( M_{max} = 1.4 \). The velocity magnitude distribution in the male rotor-housing gap is shown in Fig. 9. The static pressure distribution along the centerline of the clearance gap is visualized in Fig. 10.

**Conclusions**

The original results computed using the numerical code developed by the author demonstrate that the leakage flow through the 2-D model of the 100 \( \mu \text{m} \) wide male rotor-housing gap is transonic for both prescribed pressure ratios \( p_{in}/p_2 = 2 \) and \( p_{in}/p_2 = 3 \) but without shock waves typical for transonic flows, [3], at macroscales. Probably, that is a result of the viscous fluid flow in the very narrow channels. A key non-dimensional parameter for gas microflows is the Knudsen number. According to [5] it is related to the Reynolds and Mach numbers as follows \( Kn = (M/Re)\sqrt{\pi/2} \). It can be easily shown that our computations lead to \( Kn < 5 \times 10^{-4} \), i.e. the fluid can be considered as a continuum and the application of the mathematical model based on the system of the compressible FANS equations is acceptable in our case. The presented numerical method combined with the algebraic Baldwin-Lomax turbulence model is able to work also on the computational grid refined near the walls without numerical oscillations.

---

![Fig. 3](image_url)  
**Fig. 3** Isolines of the Mach number distribution in the 2-D model of the clearance gap for the pressure ratio \( \overline{p}_1/\overline{p}_2 = 2 \)

![Fig. 4](image_url)  
**Fig. 4** Velocity magnitude distribution \([\text{m/s}]\) in the 2-D model of the clearance gap for the pressure ratio \( \overline{p}_1/\overline{p}_2 = 2 \)
Fig. 5  Isolines of the static pressure \([\text{Pa}]\) (up) and static pressure distribution along the centerline of the 2-D model of the clearance gap (down) for the pressure ratio \(\bar{p}_u/\bar{p}_i = 2\)

Fig. 6  Comparison of static pressure distribution along the centerline of the clearance gap for the pressure ratio \(\bar{p}_u/\bar{p}_i = 2\)

Fig. 7  Comparison of Mach number distribution along the centerline of the clearance gap for the pressure ratio \(\bar{p}_u/\bar{p}_i = 2\)
Fig. 8  Isolines of the Mach number distribution in the 2-D model of the clearance gap for the pressure ratio \( \bar{p}_u/\bar{p}_1 = 3 \)

Fig. 9  Velocity magnitude distribution \([\text{m s}^{-1}]\) in the 2-D model of the clearance gap for the pressure ratio \( \bar{p}_u/\bar{p}_1 = 3 \)

Fig. 10  Static pressure distribution along the centerline of the 2-D model of the clearance gap for the pressure ratio \( \bar{p}_u/\bar{p}_1 = 3 \)

Presently, the experimental investigation of the leakage flow through this 2-D model of the clearance gap for both pressure ratios is prepared in the Institute of Thermomechanics of Academy of Sciences of the Czech Republic in Prague.

Acknowledgement

This investigation was supported by the research project MSM 4977751303 of the Ministry of Education, Youth and Sports of the Czech Republic.
References


