COMPUTATION OF THE FLOW AND INTERACTION OF SHOCK WAVES IN A 2D SUPERSONIC EJECTOR

Jan Šimák
Výzkumný a zkušební letecký ústav, Prague

1 Introduction

The aim of this work is to suggest a suitable numerical method for a solution of a flow in a two-dimensional supersonic ejector. This method could be used for a task of a shape optimization in the future, but now only remarks and first results are discussed. Since a solver plays an essential role in an optimization problem, the focus is not only the accuracy but also the speed. The presented method was tested on an experimental model of an ejector where the flow pattern is known. Dvořák, Šafařík in their work [1] presented results of experiments from an aerodynamic wind tunnel for a various flow regimes. In a work by Kolář, Dvořák [2], a discussion about possibilities of numerical solutions with the software Fluent can be found. Since the agreement with the experiment is only rough, there is an effort to improve the results.

2 Mathematical model

With respect to the properties of the considered problem, where interactions between shock waves and boundary layers are significant, it is necessary to take into account the viscosity of the flow. From that reason a system of the Navier-Stokes equations equipped with a $k-\omega$ turbulence model was chosen. This system can be written in a vector form

$$\frac{\partial \boldsymbol{w}}{\partial t}+\sum_{i=1}^{2}\frac{\partial \boldsymbol{F}_i(\boldsymbol{w})}{\partial x_i}=\sum_{i=1}^{2}\frac{\partial \boldsymbol{R}_i\left(\boldsymbol{w},\nabla\boldsymbol{w}\right)}{\partial x_i}+\boldsymbol{S}\left(\boldsymbol{w},\nabla\boldsymbol{w}\right), \tag{1}$$

where

$$\boldsymbol{w}=\left(\rho,\rho v_1,\rho v_2,E,\rho k,\rho\omega\right)^T, \tag{2}$$

$$\boldsymbol{F}_i(\boldsymbol{w})=\left(\rho v_i,\rho v_1 v_i+\delta_{1i}p,\rho v_2 v_i+\delta_{2i}p,(E+p)v_i,\rho k v_i,\rho\omega v_i\right)^T, \tag{3}$$

$$\boldsymbol{R}_i(\boldsymbol{w},\nabla\boldsymbol{w})=\left(0,\tau_{i1},\tau_{i2},\tau_{i1}v_1+\tau_{i2}v_2+\left(\frac{\mu}{P_r}+\frac{\mu_T}{P_{rT}}\right)\gamma\frac{\partial e}{\partial x_i},(\mu+\sigma_k\mu_T)\frac{\partial k}{\partial x_i},\right.$$
$$\left.(\mu+\sigma_\omega\mu_T)\frac{\partial\omega}{\partial x_i}\right)^T, \tag{4}$$

$$\boldsymbol{S}(\boldsymbol{w},\nabla\boldsymbol{w})=\left(0,0,0,0,P_k-\beta^*\rho\omega k,P_\omega-\beta\rho\omega^2+C_D\right)^T. \tag{5}$$

The state variables p,ρ,v_1,v_2 denote the static pressure, density and velocity components. Next to them, there are the turbulent kinetic energy k and specific turbulent dissipation ω. By the symbol E is denoted the total energy, by e the inner energy, μ viscosity coefficient, γ Poisson adiabatic constant, Pr Prandtl number, τ_{ij} components of the stress tensor and δ_{ij} is the Kronecker delta. The coefficient of the dynamic viscosity μ is determined by the Sutherland's relation. By a subscript T are denoted coefficients related to turbulence. The productions of the turbulent kinetic energy and the turbulent dissipation in the equations for k and ω are represented by terms P_k and P_ω. The last term C_D represents a cross diffusion. The parameters of the turbulence model are taken from Kok [3] or Wilcox [4]. The Wallin's EARSM model based on $k-\omega$ [5] is also used in the computations. The system is closed by the equation of state for a perfect gas.

The problem involves three types of boundary conditions. At inlet boundaries, there are prescribed a total pressure, total temperature and a tangential velocity. Moreover, an intensity of the turbulence and a turbulent Reynolds number need to be given. At an outlet boundary, there is prescribed a static pressure. The boundary condition for a wall implies a zero velocity, zero turbulent kinetic energy and an adiabatic wall. The other necessary variables are computed from inside of the domain.

If the turbulent kinetic energy $k = 0$, then the system separates into two parts. The turbulence model has no influence upon the Navier-Stokes equations and the laminar model is described.

3 Numerical model

The system of equations described above is solved by the implicit finite volume method. The computational domain is discretized with the use of a structured quadrilateral grid. With respect to the complexity of the geometry, the grid consists of several blocks, where the solution of the system is computed separately. Linearized equations are solved by the GMRES method [6]. The equations for k and ω are uncoupled from the N.-S. equations and thus two systems are solved. In this case, only the corresponding components of vectors are used. From the reason of a higher precision, a scheme of higher order with the application of the Van Albada limiter is used.

In the following text, discretized variables and functions are denoted by a subscript h. The scheme of a FV method at the time level t^{k+1} can be written in the form

$$\left(\boldsymbol{I} + \frac{\mathrm{D}\boldsymbol{\Phi}\left(\boldsymbol{w}_h^k\right)}{\mathrm{D}\boldsymbol{w}}\right)\left(\boldsymbol{w}_h^{k+1} - \boldsymbol{w}_h^k\right) = -\boldsymbol{\Phi}\left(\boldsymbol{w}_h^k\right), \tag{6}$$

where blocks of the function $\boldsymbol{\Phi}$ corresponding to particular grid cells are

$$\boldsymbol{\Phi}_i(\boldsymbol{w}) = \frac{\tau^k}{|D_i|}\sum_{j\in S(i)}\left(\sum_{s=1}^{2} n_{ij,s}\boldsymbol{F}_{s,h}(\boldsymbol{w};i,j)\,|\Gamma_{ij}| - \sum_{s=1}^{2} n_{ij,s}\boldsymbol{R}_{s,h}(\boldsymbol{w};i,j)\,|\Gamma_{ij}|\right) - \tau^k\boldsymbol{S}_h(\boldsymbol{w};i,j). \tag{7}$$

The symbol τ^k denotes a time step, $|D_i|$ the area of a cell, $|\Gamma_{ij}|$ the length of an edge between cells i and j, $S(i)$ is a set of indices of neighbouring cells and finally $\boldsymbol{n}_{ij} = (n_{ij,1}, n_{ij,2})$ is an outer normal unit vector. Gradients on edges are computed from values in centres of the six neighbouring cells.

Convective terms $\sum_{s=1}^{2} n_{ij,s}\boldsymbol{F}_s\left(\boldsymbol{w}\left(\bullet, t^k\right)\right)\big|_{\Gamma_{ij}}$ on the common edge of cells i and j are approximated with the use of a numerical flux $\boldsymbol{H}\left(\boldsymbol{w}_i, \boldsymbol{w}_j, \boldsymbol{n}_{ij}\right) = \boldsymbol{Q}^{-1}\boldsymbol{g}\left(\boldsymbol{q}_i, \boldsymbol{q}_j\right)$, where $\boldsymbol{q} = \boldsymbol{Q}\boldsymbol{w}$ and the matrix $\boldsymbol{Q}$ denotes a transformation of the coordinate system. In the part concerning the N.-S. equations, the approximate Riemann solver $\boldsymbol{g}\left(\boldsymbol{q}_i, \boldsymbol{q}_j\right)$ is expressed by the Osher-Solomon scheme [7]. The inlet and outlet boundary conditions are based on a direct solution of a 1D Riemann problem

$$\frac{\partial \boldsymbol{q}}{\partial t} + \frac{\partial \boldsymbol{F}_1(\boldsymbol{q})}{\partial \tilde{x}} = 0, \quad (\tilde{x}, t) \in \mathbb{R} \times (0, \infty). \tag{8}$$

In the part concerning turbulence equations, the Vijayasundaram scheme is used,

$$\boldsymbol{g}_V\left(\boldsymbol{q}_i, \boldsymbol{q}_j\right) = \boldsymbol{A}_1^+\left(\frac{\boldsymbol{q}_i + \boldsymbol{q}_j}{2}\right)\boldsymbol{q}_i + \boldsymbol{A}_1^-\left(\frac{\boldsymbol{q}_i + \boldsymbol{q}_j}{2}\right)\boldsymbol{q}_j. \tag{9}$$

For the reason of comparison, another schemes were used, too. Namely the schemes based on the Osher-Solomon method, on the direct solution of the Riemann problem or the Van Leer scheme

$$\boldsymbol{g}_{VL}\left(\boldsymbol{q}_i, \boldsymbol{q}_j\right) = \frac{1}{2}\left(\boldsymbol{F}_1(\boldsymbol{q}_i) + \boldsymbol{F}_1(\boldsymbol{q}_j) - \left|\boldsymbol{A}_1\left(\frac{\boldsymbol{q}_i + \boldsymbol{q}_j}{2}\right)\right|(\boldsymbol{q}_j - \boldsymbol{q}_i)\right). \tag{10}$$

The matrix $\boldsymbol{A}_1(\boldsymbol{q})$ is the Jacobi matrix of the function $\boldsymbol{F}_1(\boldsymbol{q}) = (\rho k v^n, \rho\omega v^n)^T$, where $\boldsymbol{q} = (\rho k, \rho\omega)^T$ and v^n is the velocity in the direction of the outer normal. Symbols $\boldsymbol{A}_1^+$ and $\boldsymbol{A}_1^-$ denote the positive and negative parts of $\boldsymbol{A}_1$, defined with the use of its eigenvalues.

A gas in a rest is chosen as an initial condition for the system. Since the aim is to get a steady state solution, the time step is gradually stretched to a given limit. It is necessary to take into account the changes in the flow field, especially in the beginning of the computation, which can lead to a breakdown of the method if the time step is too long. This can be avoided by setting some dependence between the time step and the residuum.

4 Description of the problem

The described method was used to compute a flow field in a 2D ejector (see Fig. 1). The prescribed ratio of total pressures at the inlet of the driving and driven gas $p_{01}/p_{02} = 4.09$, the ratio of total temperatures $T_{01}/T_{02} = 1$. At the beginning, the geometry of the whole ejector was considered and the results are depicted in Fig. 2. Depending on the prescribed static pressure at the outlet boundary, there is an area inside the mixing chamber where the flow is supersonic. This flow is almost symmetric with respect to the x-axis. From that reason, it is possible to deal only with the upper half of the ejector and also shorten the length of the mixing chamber. The static pressure at the outlet boundary is prescribed, in this case, sufficiently low in order to get a supersonic flow.

5 Results

The geometry of the upper part of the ejector was divided into four blocks. Each of the primary and secondary nozzles corresponds to one block and the mixing chamber consists of two blocks. At the tip of the nozzles, all four blocks connect each other. The problem was first solved on a coarse grid and after that, the grid was refined to have 500×330 cells covering the mixing chamber.

Results were obtained with various modifications of the software in order to find the best numerical method. The results with the standard Wilcox $k-\omega$ model, $k-\omega$ model according to Kok and EARSM model based on $k-\omega$ are similar and the pattern of shock waves is almost the same. Differences can be found in the thickness of the boundary layer and in the velocity profile across the boundary layer (Fig. 3).

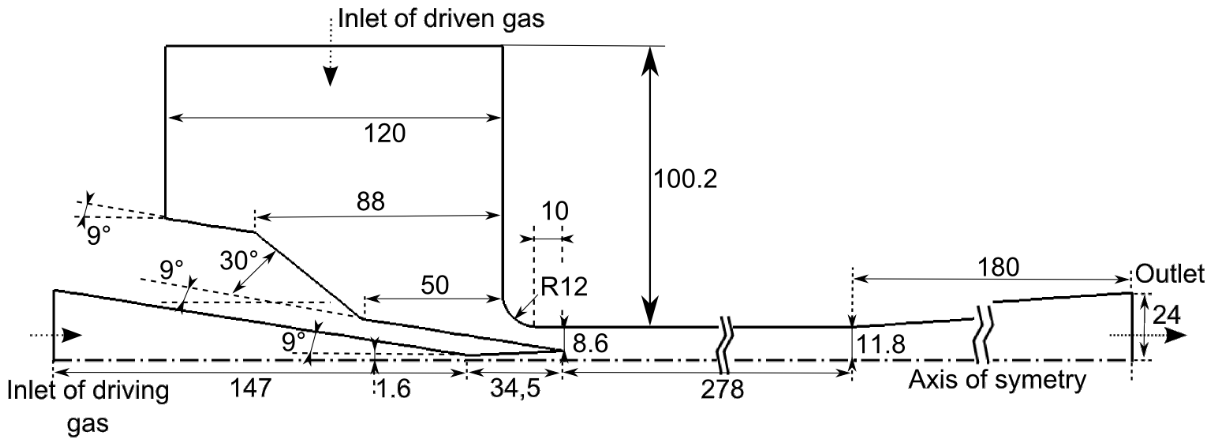

Figure 1: Model of the ejector (in milimeters)

What is also important besides the turbulence model, at least according to results, seems to be the discretization of convective terms. These influence the interior of the mixing chamber in contrast to viscous terms, which dominated only near walls or a wake. The discretizations based on Vijayasundaram, Osher-Solomon and based on a direct solution of the Riemann problem give similar results. On the other hand, the discretization based on the Van Leer scheme gives different results. This scheme seems to have a larger numerical diffusion which smears the turbulences. Some information can be obtain form the laminar model, whose prediction is good but gives unrealistic boundary layer separation. The results with the Van Leer scheme can be viewed as something between laminar and the other turbulent schemes.

In Fig. 4, there are patterns of shock waves from computations and they are compared with positions of significant points obtained from experiment [1]. Though there is not a perfect agreement, it is possible to get a reliable image of the flow field. Fig. 5 shows Mach number isolines obtained with the above mentioned modifications of the method and also some results obtained with the software Fluent (pressure-based model, quadrilateral unstructured grid) [2]. The comparison with Fluent is only rough, because it is necessary to unify grids and parameters of the model.

6 Conclusion

The above presented results show that it is possible to get a reliable prediction of the flow in a supersonic ejector using a numerical method. The implemented $k - \omega$ turbulence model seems sufficient to capture all important information about the flow. It is necessary to do a thorough analysis of boundary and shear layers, at this time the model based on Kok turns out to be a good choice. As for the discretization of convective terms, the Osher-Solomon scheme gives comparable results with the direct solution of the Riemann problem, but is cheaper. In the turbulence part, the scheme based on Vijayasundaram is comparable to Osher-Solomon and the direct solution of the Riemann problem, which are more expensive. The results from the Van Leer and laminar versions indicate that some interesting results can be expected from a modelling of a turbulent transition. It is necessary to do a thorough analysis of the method and further comparison of results.

Acknowledgements This work was supported by the Grant MSM 0001066902 of the Ministry of Education, Youth and Sports of the Czech Republic.

References

[1] Dvořák V., Šafařík P.: Supersonic Flow Structure in the Entrance Part of a Mixing Chamber of 2D Model Ejector; J. of Thermal Science Vol.12, No.4, 2003

[2] Kolář J., Dvořák V.: Interaction of shock waves in supersonic ejector; proceeding, Setkání kateder mechaniky tekutin a termodynamiky, Plzeň, 24.-27. 6. 2008

[3] Kok J. C.: Resolving the Dependence on Freestream Values for the $k - \omega$ Turbulence Model; AIAA Journal Vol. 38, No. 7, July 2000

[4] Wilcox D. C.: Turbulence Modeling for CFD, 2nd ed.; DCW Industries Inc., 1998

[5] Wallin S.: Engineering turbulence modelling for CFD with a focus on explicit algebraic Reynolds stress models; doctoral thesis, Royal Institute of Technology, Stockholm, 2000

[6] Saad Y.: Iterative Methods for Sparse Linear Systems, 2nd ed.; SIAM, 2003

[7] Feistauer M., Felcman J., Straškraba I.: Mathematical and Computational Methods for Compressible Flow; Clarendon Press, Oxford, 2003

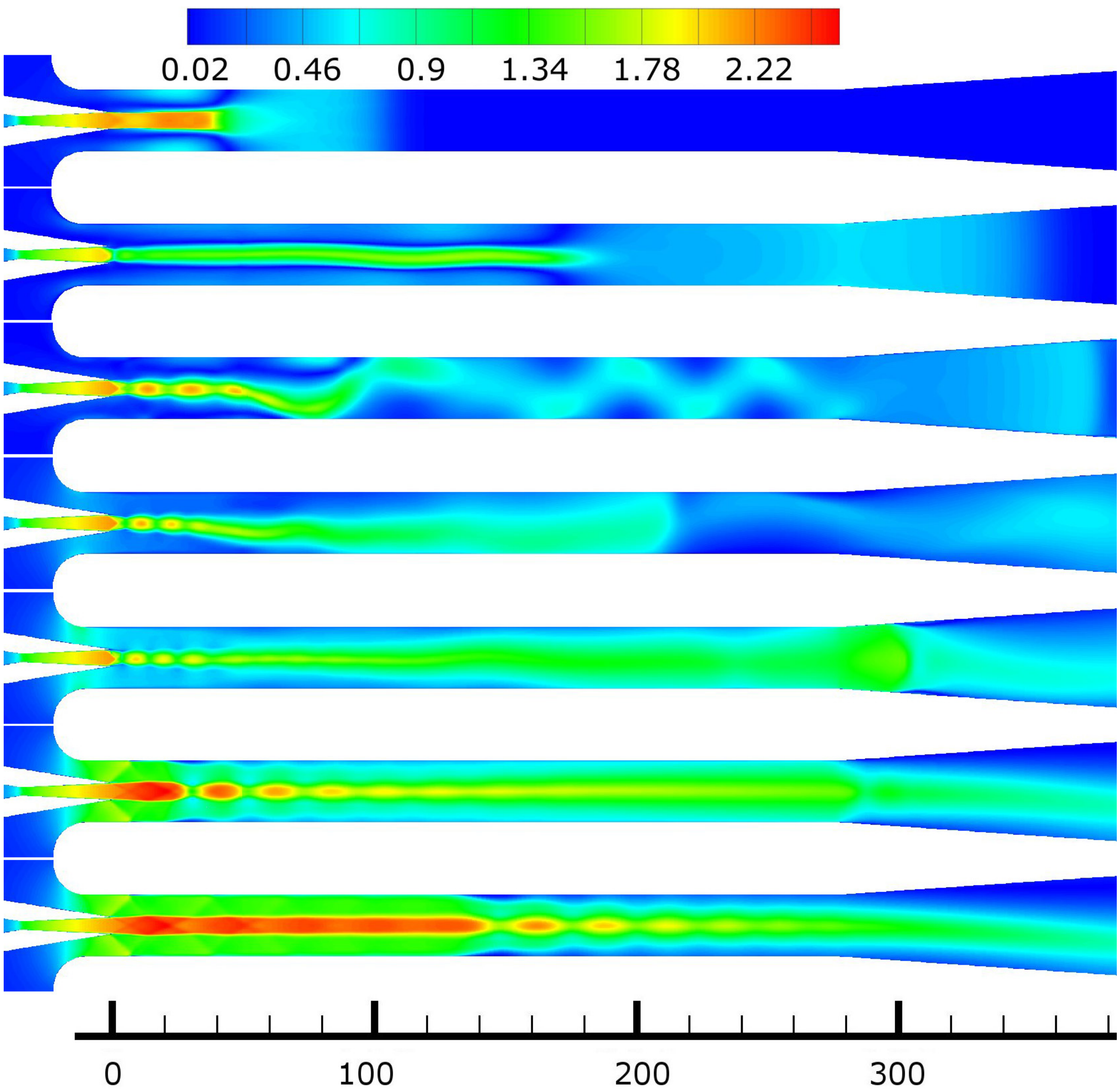

Figure 2: Evolution of the flow in the ejector (selected iterations), Mach number distribution, $p_{01}/p_{02} = 4.09$, $p_{02}/p_b = 1$, $k - \omega$ model (Kok)

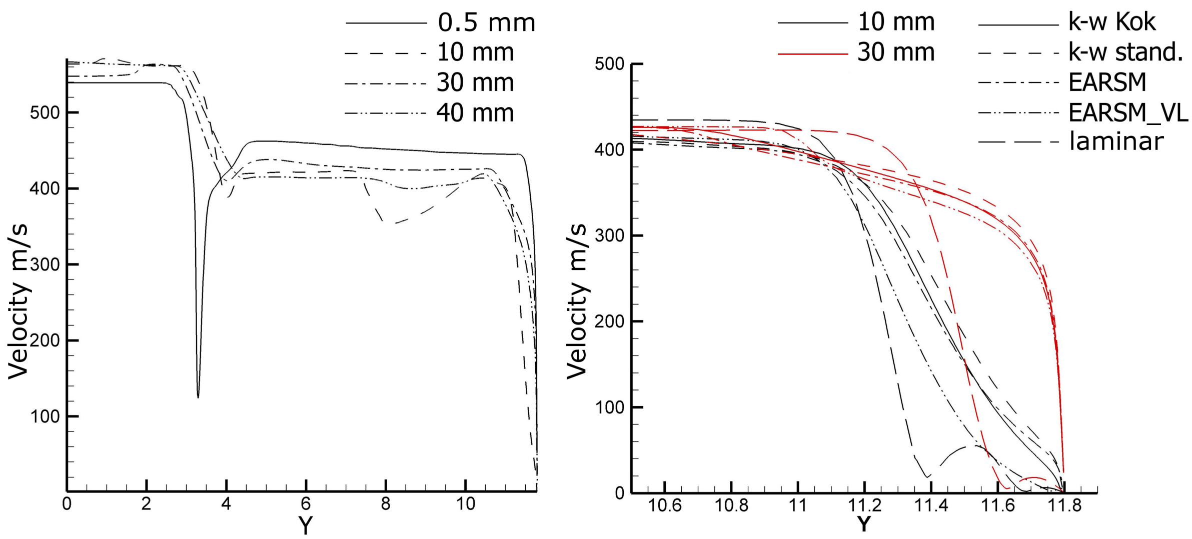

Figure 3: Velocity distribution across the mixing chamber at different points, on the left for $k - \omega$ (Kok) and on the right for various models

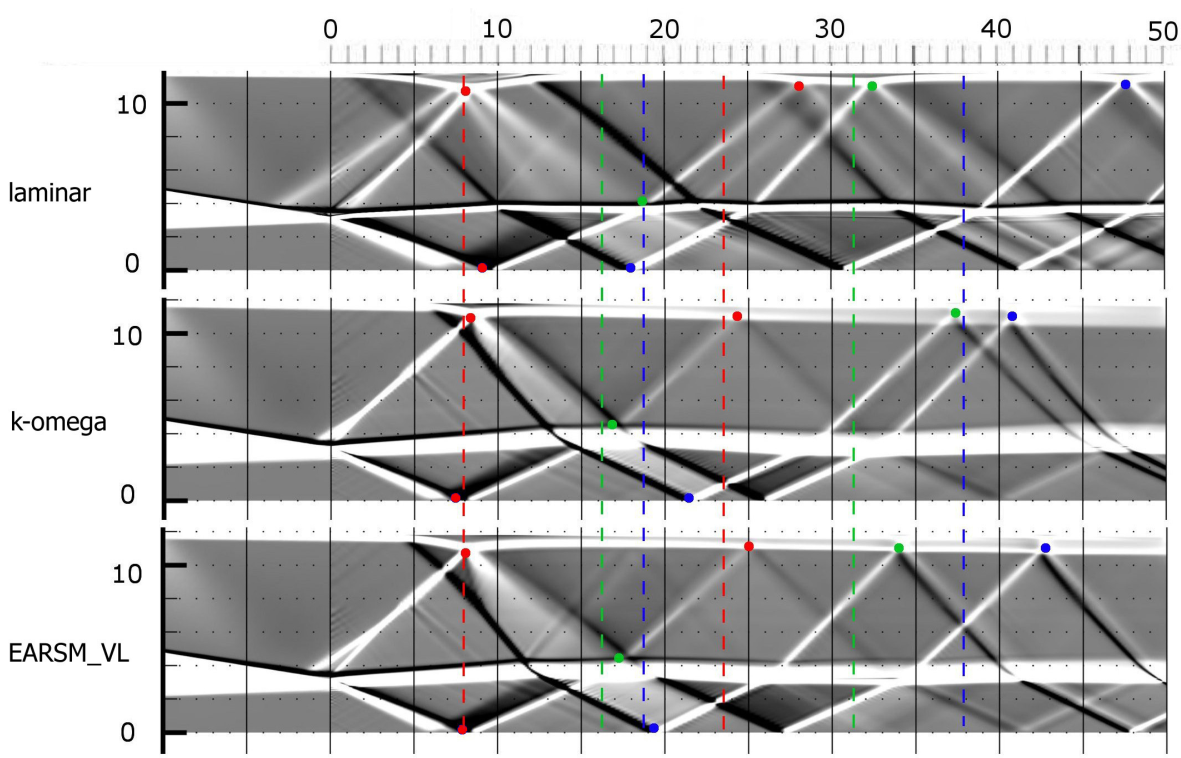

Figure 4: Derivative of density in vertical direction, obtained by computation. The first example is a laminar flow, second $k - \omega$ (Kok, Vijayasundaram scheme int turb. model), third EARSM (Van Leer scheme in turb. model). Dashed lines show positions of significant points from experiment (Dvořák V., Šafařík P. [1]).

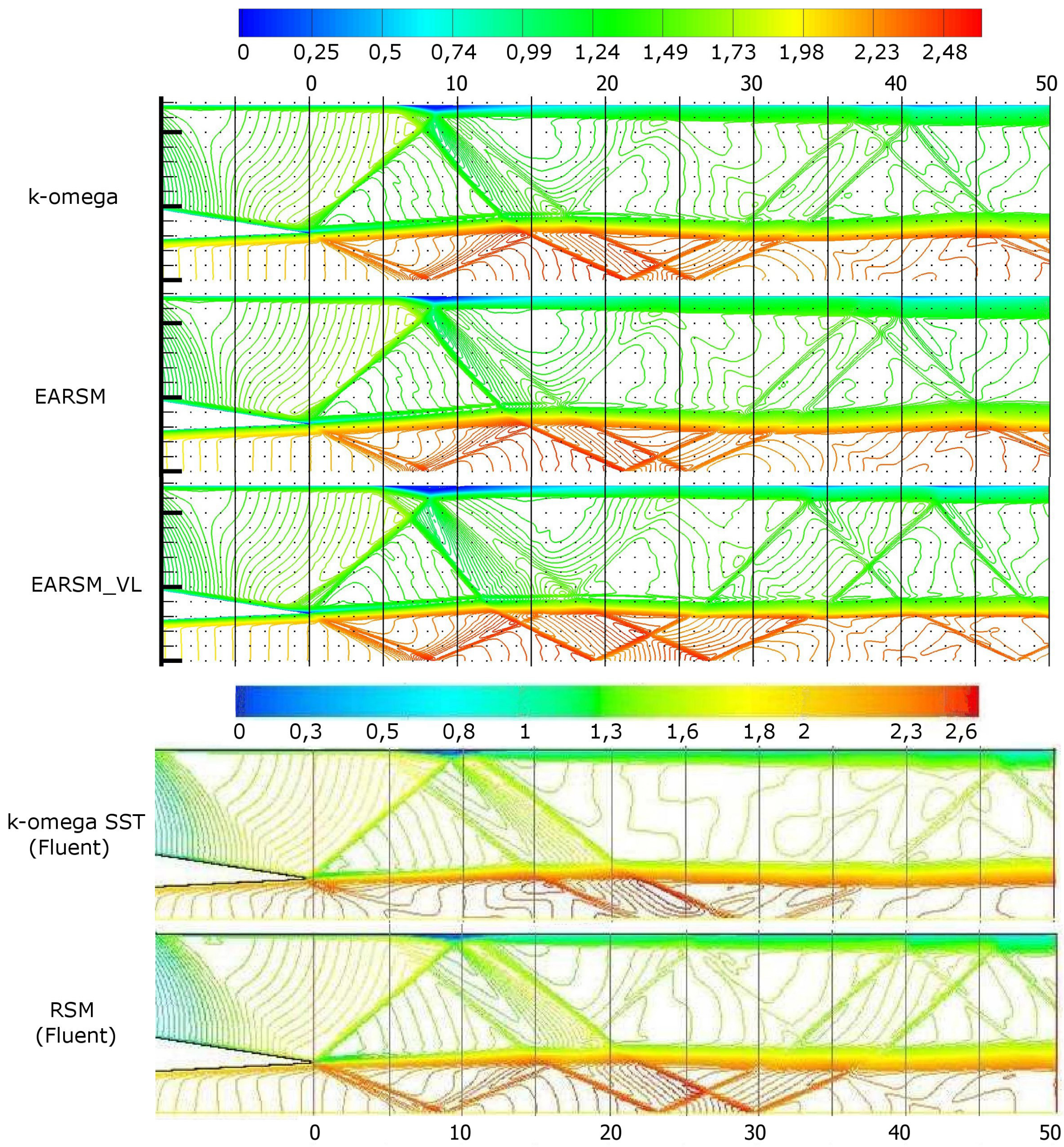

Figure 5: Mach number distribution (results with different models, Fluent [2])