Computer Assisted Mechanics and Engineering Sciences, **6**: 251–261, 1999.

Numerical modelling the air flow in parts of air jet loom

Karel Adámek
VÚTS – Research Institute for Textile Machines Liberec
U jezu 4, 46119 Liberec, Czech Republic

(Received March 8, 1999)

Paper summarizes the first results of two-dimensional (2D) numerically modelled expansion and flow of compressible and non-viscous gas in typical parts of air jet weaving system; namely in main nozzle designed as an ejector with various shapes of the mixing zone, in relay (auxiliary) nozzle with substantial flow separation in the rash flow bend directly before the nozzle outlet, and the influence of the reed dent edges shape on the free stream reflection and penetration through reed gaps along a real "porous" wall.

The used Euler's equations are solved by a Finite Volumes Method (FVM) with automatic mesh generation and optimization of unstructured triangle mesh. Graphical results show 2D isolines of all gas state values, further Mach number, entropy and velocity vectors. 1D profiles of all quantities along chosen cross-sections or surfaces can be obtained, too. They give to the designer a large and quick review about the problem.

The coincidence with experiment, measuring and real weaving tests is very good. The advantage of numerical modelling consists in the very quick, simple and user-friendly operation.

1. INTRODUCTION

Weaving technology is one of the oldest production systems but the principle is always the same — see Fig. 1: The longitudinal system of warp ends (1) is periodically opened and in the arising space (2) is transversally inserted filling end (3). The fabric (4) arises by gradual changing the warp ends opening and beating-up by the reed (5) of inserted filling ends...

Air jet weaving system, developed in 50's in VÚTS Liberec and spread all over the world, allows the highest weaving performance. Weft transport in the air jet system is realized by aerodynamic forces of flows from the main nozzle (6) and of system of relay nozzles (7), situated along the reed channel (8).

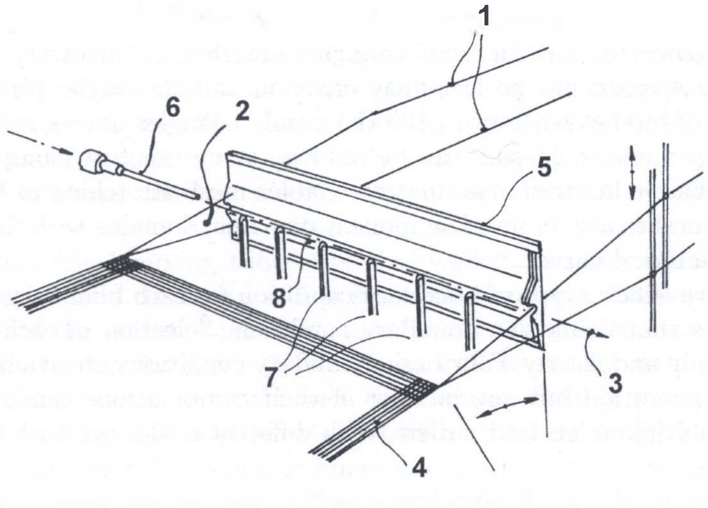

Fig. 1. Scheme of air jet weaving system

As an example, modern air jet loom can operate at 1500 r.p.m. at the standard fabric width of 1.9 m. When the weft picking time is of about 50 per cent of each working period, then the medium picking velocity reaches the value of 85 m/s (300 km/h) approx. In addition, the velocity field arises again in each working period because it is periodically abolished during beat-up motion of mechanism.

To get a quick, reliable and economic air jet weft insertion it is useful to study aerodynamic flowfield on the loom [1, 2, 3, 5, 7]. By contemporary development of fundamental parts of air picking system the numerical modelling is applied. There the internal flow and expansion of compressed air in weaving nozzles are studied — both the main and auxiliary ones — as well as the free stream propagation in the atmosphere leaving the nozzle outlet with interaction between the stream and complicated reed wall–partial flow reflection and penetration. Obtained velocity (dynamic pressure) field transports the textile yarn — the linear body with small and bad defined cross dimensions and slight bend stiffnes — to create the cloth.

Aerodynamic forces influencing on acceleration and transport of weft yarn are limited with dissipation of free stream going out from nozzle outlet. For increase of the weaving performance and reliability it is necessary to eliminate the dissipation in questuion by several means. Analysis of essential parts of air picking system is closed with numerical 3-dimensional model of viscous turbulent flow of expanding air in weaving channel of complicated shape with partially "porous" walls.

2. THE USED METHOD

The used software [6] is a Windows application for calculations of two-dimensional compressible inviscid flows. Graphically oriented interface together with a robust numerical method and the computational mesh optimization enable the user to simulate most of two-dimensional fluid dynamics problem, which can be described by the Euler's equations. The program uses unstructured triangular meshes to discretize a computational domain and therefore has no limitation on the domain shape. Tasks with multiple inlets and outlets, as well as periodical problems can be solved. The flow can be stationary or unsteady, with shock waves or without them, subsonic, transonic or supersonic. However the fluid should have properties of the ideal gas and the flow is supposed to be isentropic and not significantly influenced by viscous effects or turbulence.

The aim of pre-processing is to input geometrical data of a computational domain, discretize the domain and input boundary conditions and other paramaters for the calculation.

Module Geometry designs boundary curves of a computational domain. The program automatically analyses the structure of given objects consisting of any number of poly-lines, arcs, circles and/or cubical splines. More complex lines can be read from a file.

Module Meshgen generates unstructured triangular meshes for arbitrary two-dimensional domains. The algorithm is based on the Delaunay criterion and thorough optimization of this code allows the generation of meshes with up to 100 thousands triangles during several minutes on PCs with 486 or Pentium processors. Meshes can be refined at any point or along any curve in the solution domain and a simple linear transformation enables mesh stretching in the desired direction. Module can generate meshes also in multicomponent domains, domains with "holes" or single points and in domanis with internal curves.

Module Boundary specifies a type of boundary condition for each boundary curve or its part and values of all quantities that define the boundary condition. Selection of each boundary condition range is made very easily and clearly. Four basic boundary conditions are available: solid wall, inlet, outlet and periodical condition but any number of their combinations can be used. This enables to specify flow with multiple inlets and outlets, with different outlet pressure values, different inlet flow angles etc.

The finite volume method with TVD-schemes of first and second order accuracy is used for the calculation. These schemes provide very efficient and reliable solver. Results of any previous calculation of similar geometry can be interpolated into different unstructured mesh and used as initial condition for next calculation. There are virtually no limitation on the number of mesh triangles. In practice, however, meshes with 5–20 thousands triangles seem to be optimal for PCs with Pentium processors and give very precise numerical solution, especially when the mesh is optimized according to the intermediate results (for a stationary flow). Solver supports multitasking, i.e., large task can be running for several hours while other applications (for example word processing) can run simultaneously. The application has been tested on a number of classical fluid dynamics problems and its results were in a good agreement with both experimental data and other theoretical results.

Data post-processing presents results of the numerical simulation by means of contour maps, spectral maps and velocity vectors. The following quantities can be displayed: state values (pressure, temperature, density) and these of velocity, Mach number and entropy, too. The user can choose from many options to display data in the most appropriate way. Graphs of all quantities along boundaries, as well as along any selected cross-section, can be readily obtained and the data can be saved in a file. Exporting the data in any spreadsheet it is possible to express many other needed physical quantities, as for instance components (w_x , w_y) and direction ($\arctan(w_y/w_x)$) of the velocity vector, air drag proportional to ($\rho \cdot w^2$), flow density ($\rho \cdot w$), vorticity proportional to ($\mathrm{d}w_y/\mathrm{d}x - \mathrm{d}w_x/\mathrm{d}y$) etc.

3. PRESENTATION OF RESULTS

Numerical modelling gives more information about the velocity/pressure distribution in observed areas of air jet weaving system namely the internal flow of expanding air together with the free stream propagation in the atmosphere after nozzle outlet.

In general, two types of weaving nozzles are used:

- main nozzle designed as an ejector overcomes the resistance of filling premeasured and prepared on the feeder drum, accelerates and transports it into the picking channel,
- auxiliary (or relay) nozzles designed simply as a hollow needle with orifice in lateral wall are located in regular pitches along the picking channel made from thin metallic sheets positioned across to the flow direction, and keep the values of the air flow velocity at level needed for quick and sure weft transport.

3.1. Main nozzle

The main nozzle designed as an ejector accelerates the weft yarn and transports this through the reed channel. Numerical modelling can explain some complicated processes in flow of expanding compressed air along the mixing tube of an ejector. Influence of the mixing tube shape of an ejector and intensity of shock waves in the flow shows following serial in Fig. I. All dimensions of ejectors are identical, the output diameter is changed, only (with mixing tube a) cylindrical, b) stepped, c) divergent). Thermodynamic conditions are identical, too, the used pressure ratio is overcritical of $p/p_0 = 1/7$.

In Fig. Ia with cylindrical mixing tube it is observable a considerable shock wave area with non-homogenous velocity field. Intensity and position of visualized shock wave depends here first of all on inner shape of the mixing tube beginning. In Fig. Ib it is one strong shock in the step, only. This design is better for the stable weft position in flow and for its more reliable picking — the maximum flow velocity operates on the weft tip, the very flexible weft yarn is pulled out, not pushed like after Fig. Ia. The idea has been verified with actual weaving tests. Figure Ic shows the divergent shape of the mixing tube, which complies better with conditions for the used overcritical expasion. The flow field with higher velocity values and feeble shock waves was reached, only. The corresponding lengthways velocity profiles along the axis of the mixing tube are shown in Fig. 2.

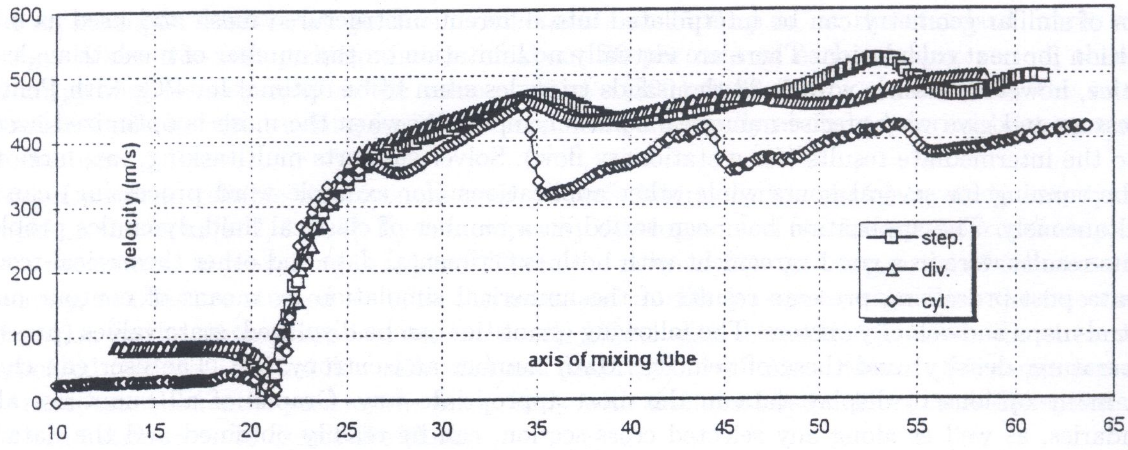

Fig. 2. Velocity profiles in ejectors from Fig. 1

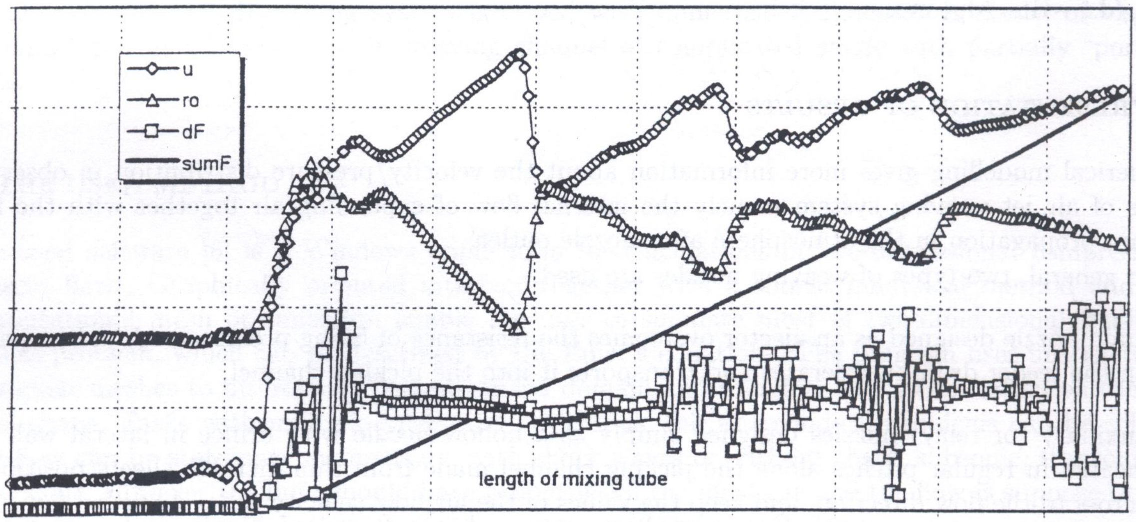

Fig. 3. Profiles of several flow quantites (in ejector from Fig. 1)

Next, Fig. 3 shows one possible evaluation of results from Fig. Ia — there are in different scales the values of u – velocity, ρ – density, $(\rho/2 \cdot w^2 \cdot \Delta x)$ – expression proportional to the elementary air drag and summary drag from the nozzle beginning.

Figure II shows detail of the central tube outlet for another design of ejector (for instance when the mixing tube is convergent or the central tube is shifted more to the left side of the design or the entry of compressed air is increased etc.). The mixing tube is here "overfilled" by the flow, some part of expanding air flows back and the necessary suction of broken weft yarn is none. Such kind of ejector is not suitable for practical weaving.

The so-called "classical" design of main jet is presented in the following Fig. III. In comparison with previous modern design the expansion is "divided" here into two parts. The first part of expansion is realized through cross wall at the entry, without contact to accelerated and transported filling, located in the middle tube. The second part of the expansion, only, has an important influence on the filling. In comparison with previous type of main nozzle, this type does not reach the highest weaving performance but at the same time it handles gently the filling. This is important for weaving materials with small firmness, as for example spun yarns. On the other side, with previous type of main nozzle after Fig. I we can reach the highest weaving performance but extreme tensile strength needs special materials, as for instance polyester rayons.

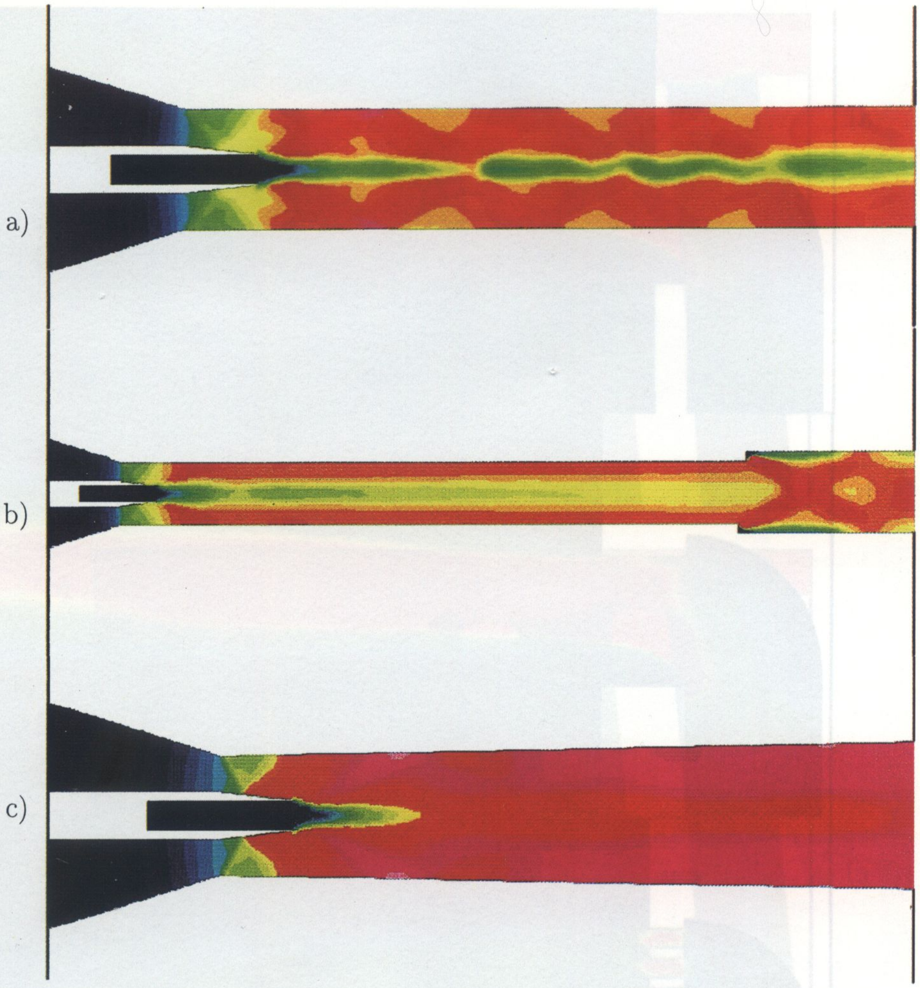

Fig. I. Velocity field of an ejector ($p/p_0 = 1/7$) with mixing tube; a) cylindrical, b) shaped, c) divergent

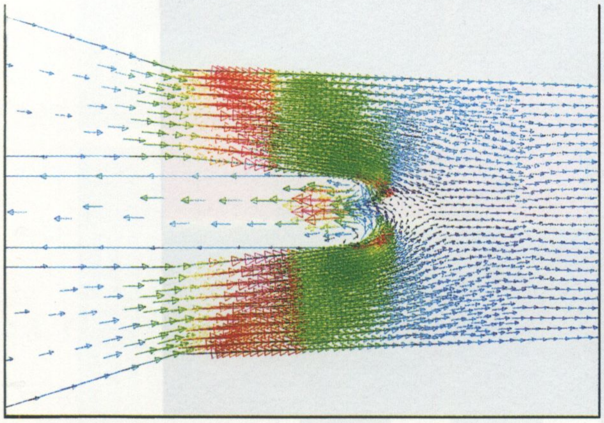

Fig. II. Backflow in the middle tube of an ejector

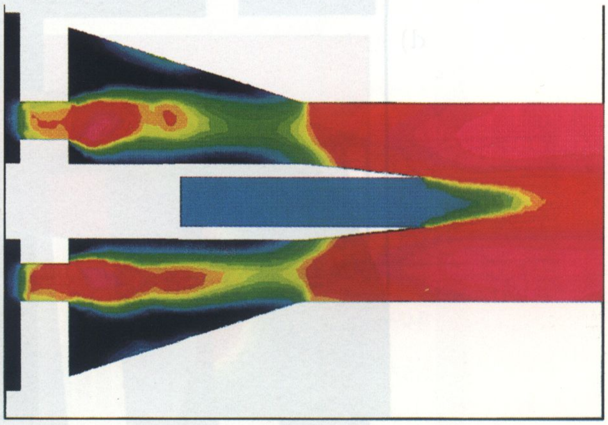

Fig. III. Detail of double expansion in the entry part of classical ejector

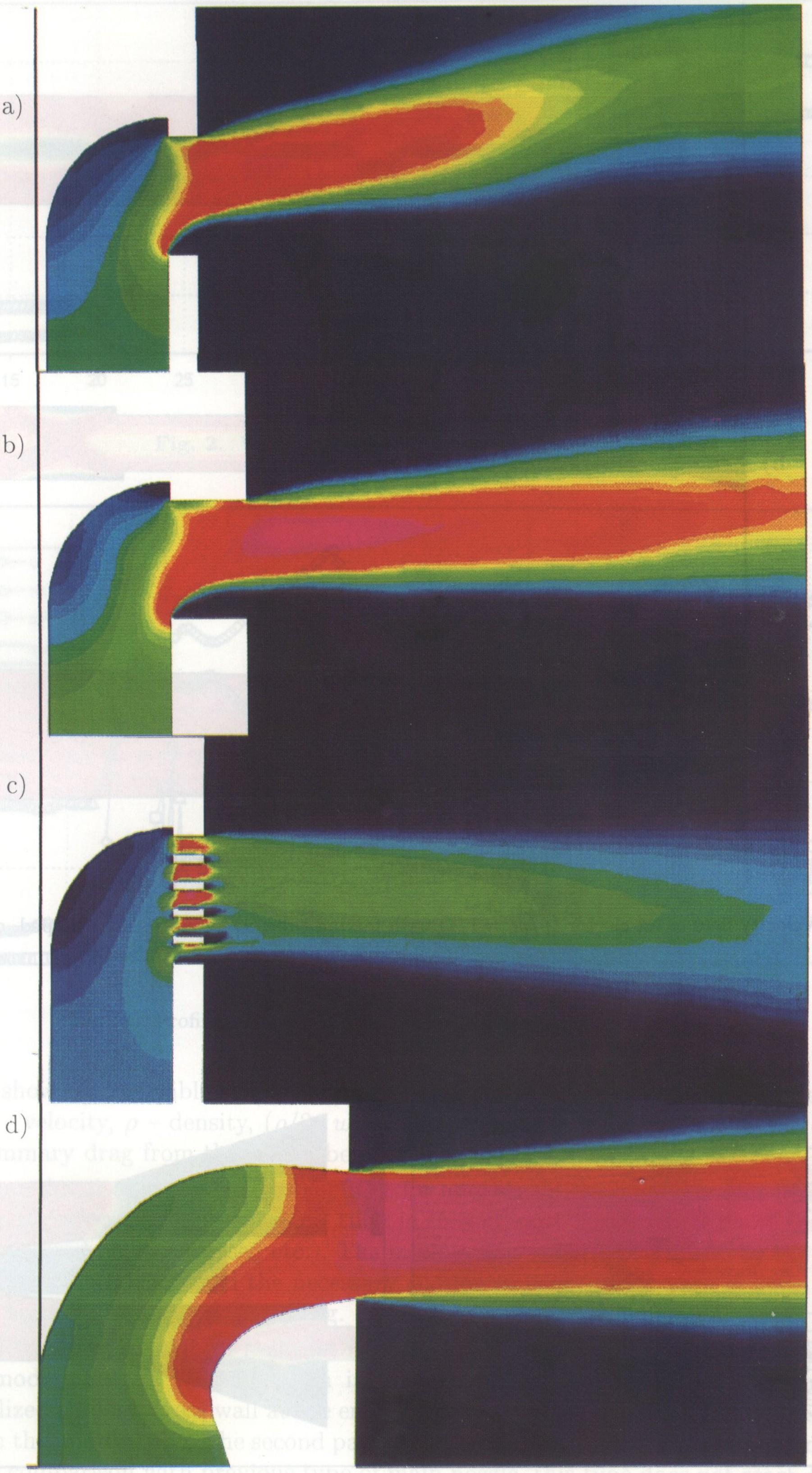

Fig. IV. Flow in outlet of several auxiliary nozzles; a) simple wall thickness, b) double wall thickness, c) multi-orifice nozzle, d) with inner channel

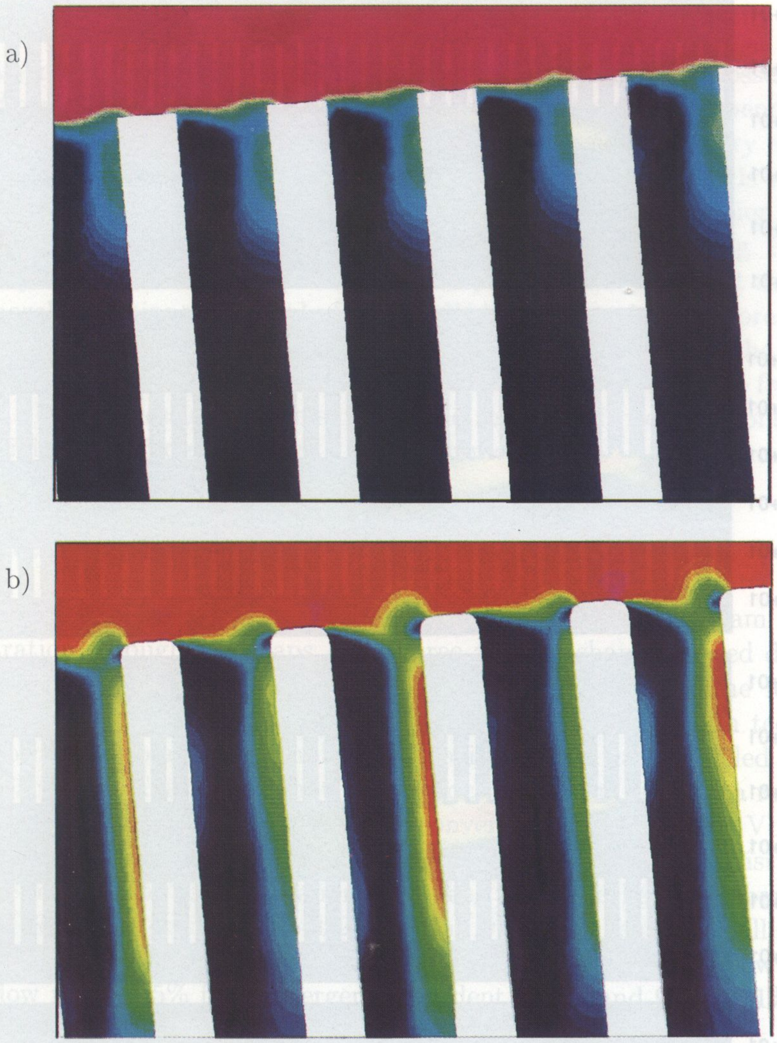

Fig. V. Flow reflection and penetration along reed wall; a) rectangular shape, b) rounded shape

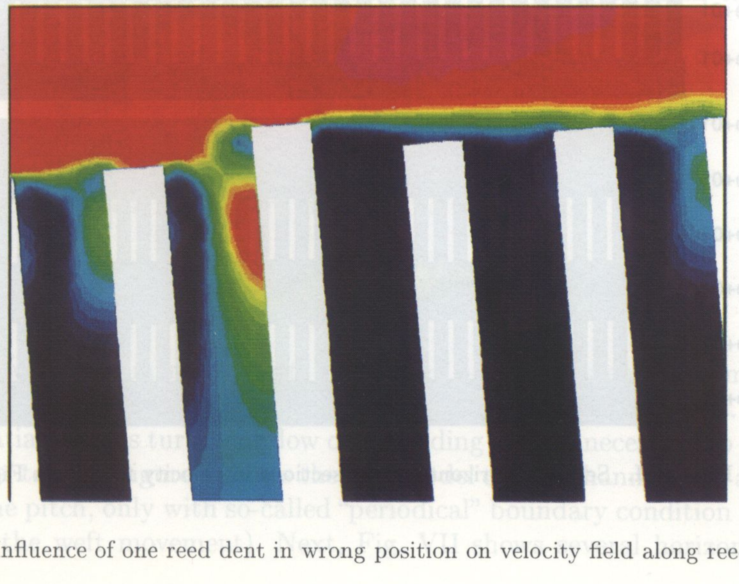

Fig. VI. Influence of one reed dent in wrong position on velocity field along reed wall

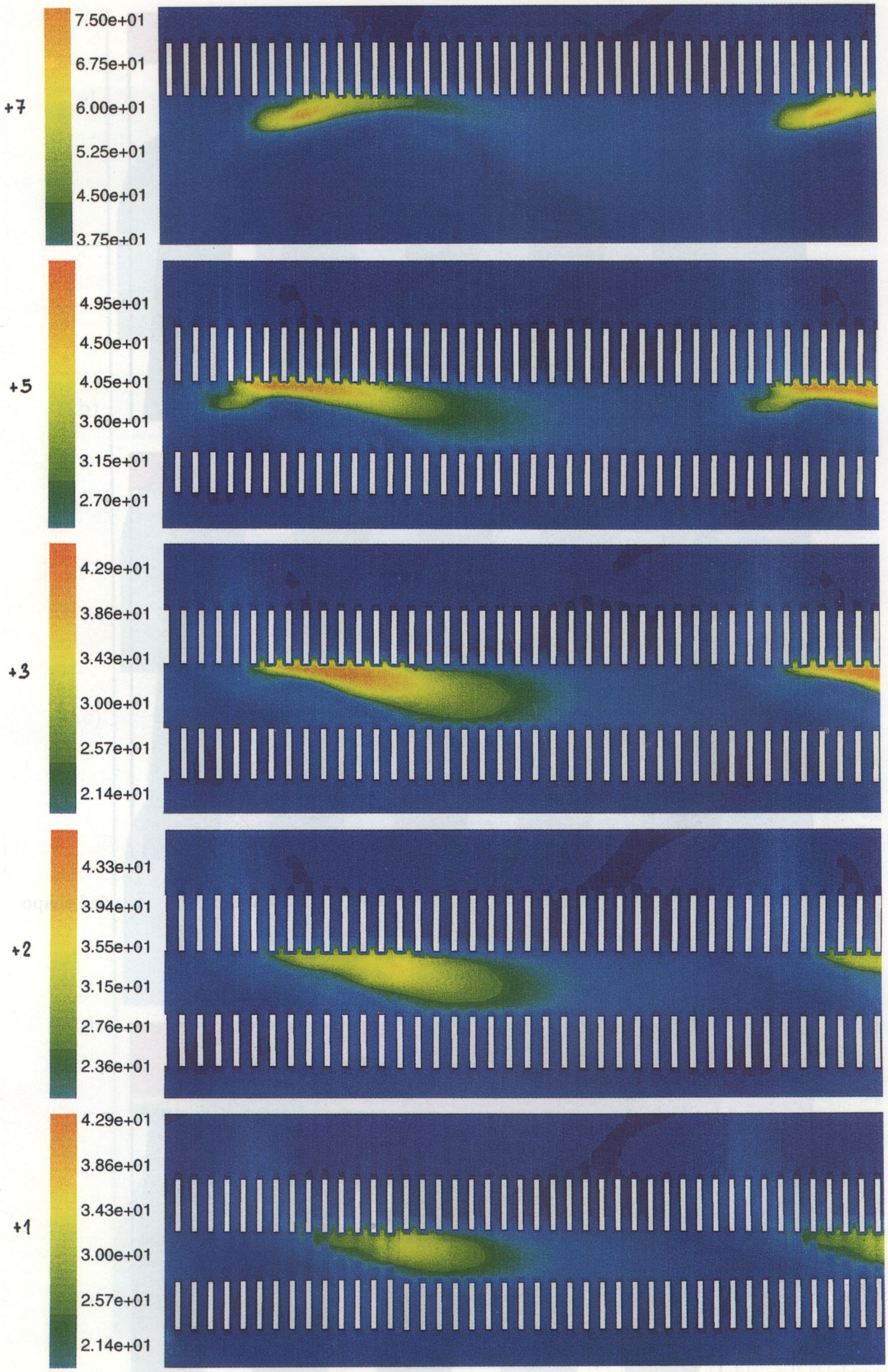

Fig. VII. Serial of horizontal cross sections in velocity field from Fig. VI

3.2. Auxiliary nozzles

System of auxiliary nozzles situated along the reed channel covers the flow losses given by free flows dissipation. The flow inside auxiliary nozzle is characterized with its sharp bend directly before the outlet. The expressive stream separation and contraction in classical auxiliary nozzle after Fig. IVa affects the unstable free stream direction, when air pressure in the supply is reset. In the greater wall thickness after Fig. IVb the flow is better guided, the free stream direction is more stable. The same positive effect has the "shower" nozzle with several small orifices, see Fig. IVc. The elementary flows create quickly one common flow, practically independent on the outlet shape. A special case is the "channel" nozzle with inner channel after Fig. IVd. Its fluent bend before the nozzle outlet is designed in such a way, it is no flow separation and the value of velocity in the outlet cross-section is approximately constant. It is ideal case, the flow is well guided and the free stream direction is independent on the air pressure in the supply. The theoretical presumptions were verified with measuring the serial of ceramic nozzles with inner channel.

3.3. Flow interaction with reed wall

The shape of reed dent edges influences on the reflection of the free stream from the reed wall and on the penetration through reed gaps, too. Three angular shapes of reed dents were modelled (rectangular, convergent and divergent) and three rounded shapes (on the left, right and both edges of each dent). The real shape of reed dents, depending on production technology, is usually convergent and both-sides-rounded. Complicated spacial reality was modelled in two dimensions, only. Typical interaction of a large constant flow (velocity of 100 m/s) with a reed wall (inclined of 5 degrees to the flow) show Fig. Va for rectangular convergent edges and Fig. Vb for rounded edges. The flowfield is similar for all tested dent shapes. The large main flow is disturbed a little, only — till the distance of the dent thickness approx. In each gap it is the flow separation accompanied with a vortex. The vortex length is of about 25% of the whole gap length for all right-angled shapes and of about 75% for all rounded shapes of reed dents. The share of the flow penetration in the whole incoming flow reaches 5% for convergent reed dent shape and 9% to 11% for all other reed dent shapes.

The very interesting situation, when one of reed dents is modelled in wrong position, shows Fig. VI:

- in the first gap it is a standard flow,
- in the second gap it is a greater flow (next reed dent is pushed out from the line of the reed wall),
- in the third gap it is a small backflow,
- in the fourth gap it is a small flow in the right direction,
- in the fifth and next gaps it is a standard flow again.

3.4. Complex view

Previous analyses of several elementary parts on air jet picking system were made as numerical models verified with experiments. Following step of the research is the synthesis.

For solving real spatial viscous turbulent flow of expanding air it is necessary to use more powerful hardware and software, too [4]. Figure 4 shows the first model of reed channel with auxiliary nozzle — the surroundings of one pitch, only with so-called "periodical" boundary condition at inlet and outlet sides (considering to the weft movement). Next, Fig. VII shows several horizontal cross sections

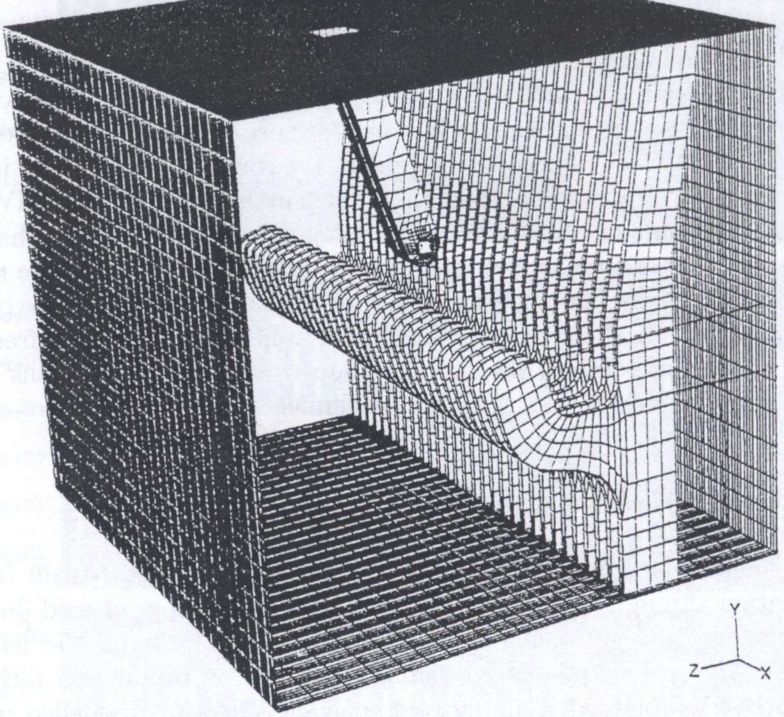

Fig. 4. Spacial model of reed channel with auxiliary nozzle

through the space of reed channel made in distances of 1–2–3–5–7 mm from channel bottom. The visualized area of free flow from the nozzle outlet presents area of maximum velocity and its partial reflection and penetration, too. Comparison of several similar solutions can express the influence as follows:

- setting or adjusting auxiliary nozzle,
- shape of the profile reed dents
- density of profile reed wall etc.

After images of velocity field we can judge about the system configuration advisable for quick, reliable and economic weft insertion. The best verification of those theoretical conclusions are given indeed with real weaving test, only, as the yarn qualities cannot be numerically modelled up to this time.

4. CONCLUSION

The aim of presented FVM application is concentrated on three points:

1. the form, position and intensity of the shock wave area, arising in the mixing tube of an ejector during the expasion from overcritical pressure ratio,
2. the flow filed in relay nozzles with considerable flow separation and free streams from relay nozzle outlet,
3. interaction of the free flow with reed wall, the flow reflection and penetration.

The used calculs replace expensive, long-term and sometimes impossible experiments as measuring, visualization etc. They allow to evaluate more variants of inner shape of nozzle before the designing. The coincidence of numerical solution with experiment or real weaving test is very good. The presented simple software gives very good qualitative imagination about the problem — it is a preference of finite elements methods in general.

The presented application of numerical modelling of two-dimensional compressible inviscid flow gives several direct and quick results for practical use by designing the weaving air nozzles:

- good imagination about the whole 2D flow (the state values of gas, position, intensity and form of shock waves) in many designed variants,
- entry data for following calculation of equation of textile yarn motion in precalculated velocity/density field,
- setting conditions for reliable nozzle operation in the weaving mill, as for example sure suction of ejector.

Real flows are naturally more complicated, furthemore by the stream expansion it should be take into account the flow viscosity and turbulence, too.

REFERENCES

[1] K. Adámek. *Free Flows* (in Czech). Research report VÚTS, Liberec, 1989.

[2] K. Adámek. Stafettendüsen hergestellt im Feingussverfahren. In: *6. Weberei Kolloquium*, Inst. für Textil- und Verfahrenstechnik, ITV Denkendorf, 1990.

[3] K. Adámek. Channel relay jets. In: *Textile Technology Int.*, 140–143, Sterling Publ., Ltd., London/Hong Kong, 1993.

[4] P. Kolář. *3D Flow in Reed Channel.* Research report for VÚTS. TechSoft Engineering, Praha, 1996.

[5] V. Kopecký, K. Adámek. Using of LDA and hydraulic analogy in air jet weaving. In: *Flow Visualization V. Proc. of the 5th Symp. on Flow Visualization, Prague, 1989*, 703–710. Hemisphere Publishing, Co., New York, 1990.

[6] M. Šejna. *Euler-2D. PC-Progress software.* Praha, 1994.

[7] R. Sintani, I. Donjou, K. Chikaoka, A. Okajima. Air jet issued from sub-nozzles in air jet looms, Part I. *JTMS Japan*, **47**: T190–T196.