PERFORMANCE OF AN AIR-AIR EJECTOR: AN ATTEMPT AT NUMERICAL MODELLING

WOJCIECH SOBIESKI

University of Warmia and Mazury,
Division of Mechanics and Machinery Construction,
Oczapowskiego 11, 10-736 Olsztyn, Poland
Wojciech.Sobieski@uwm.edu.pl

(Received 25 November 2002; revised manuscript received 9 February 2003)

Abstract: The paper describes a numerical attempt to model the operation of an air-air ejector by means of a commercial numerical code. The obtained results are compared with the results of a laboratory experiment – particularly in relation to the diffuser of the modelled ejector. The substantial discrepancies between the performance characteristics obtained in the laboratory experiment and by numerical modelling are interpreted as a result of the absence of appropriate models of turbulence and mixing in the numerical code used. The paper also presents additional numerical simulations that emphasize the importance of unsteady phenomena (bifurcations of working fluid jet, turbulent mixing at the boundary of components) in the modelling of ejector equipment.

Keywords: fluid mechanics, ejectors, numerical calculations

1. Introduction

Reading the literature on ejector equipment, one may be under the impression that they are outside the scope of interest of numerical fluid mechanics. This is due to the absence of any calculation examples – practically no item relating to this problem could be found in the literature available to the author. Of course, this is not to say that this type of equipment is totally ignored by the researchers. On the contrary, Polish and foreign literature is abundant with papers on various aspect of design, performance and operation of ejectors. This situation leads to one conclusion – there must be some difficulties in numerical modelling of ejectors. Because the best way to get an insight into a problem is to conduct your own numerical experiment, the author has decided to choose this method. Since the available literature does not include indications or recommendations on how to model ejector equipment, the experiment has been based on one of the professional numerical codes, and calculations have been made using standard calculation models, well proven for other types of equipment.

2. Laboratory experiment

In order to verify the correctness of numerical simulation, it is necessary to conduct an exemplary laboratory experiment. In this case, it was the experiment described in the paper *Influence of the gas mixing level in the mixing chamber of the subsonic ejector on the diffuser performance indicators* [1]. The paper includes a sufficiently detailed description of the examined ejector geometry, the performance parameters used during the experiment, and the obtained results. This experiment has been chosen because of the simple physical flow model – of the air-air type – and the level of detail of the data contained therein.

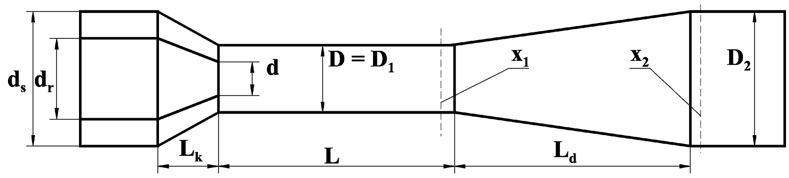

Figure 1. A diagman of an air-air ejector according to [1]

The purpose of the experimental research was to analyze the mechanism of losses occurring during compression of the gas jet in the diffuser of a single-stage subsonic ejector and to determine the indicators and the diffuser performance characteristics at various geometrical and aerodynamic parameters of the ejector.

3. Numerical experiment

The geometry presented in Figure 1 has been used to create a computer model of an ejector (2D, axial-symmetrical and 3D) (Figures 2 and 3).

Additionally, the following assumptions have been made:

- despite geometrical differences (the confusor length is unknown), a comparison of laboratory and numerical experiments may be made on the basis of diffuser performance characteristics and indicators;
- an attempt to compare the laboratory and numerical experiments can be made even if the thermodynamic parameters of the laboratory experiments are unknown. (Authors usually have given only value intervals);
- the influence of the geometry of a hydraulic hose before the ejector on the value and distribution of thermodynamic parameters is assumed to be negligibly low;
- for the purpose of the numerical experiment, only one ejector geometry will be used, namely that in which $d = 22.0$mm and $L = 221.6$mm (in the bibliographic reference article, the authors have included the results for a few diameters of the working nozzle and lengths of the mixing chamber).

Figure 2. A three-dimensional view of the model ejector used for the numerical simulation

During numerical modelling, a two-layer structural grid (near-wall region – the inside of an area) has been used, with the total number of finite volumes of 204 280 for the 3D case, and 18 354 for the 2D, axial-symmetrical case. Because the choice of the solver 2D or 3D has not had a significant impact on preliminary results of the calculations, it has been decided to continue the calculations for the axial-symmetrical geometry (2D), which has considerably reduced the time needed to achieve a satisfactory convergence.

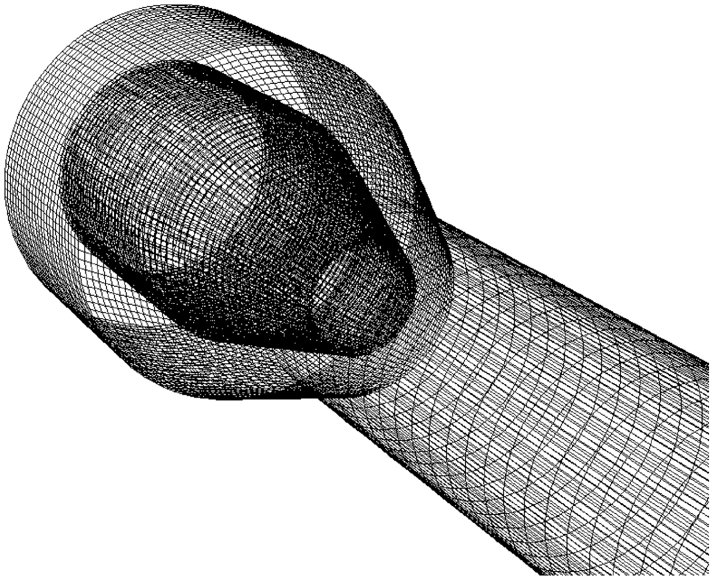

Figure 3. A fragment of the calculation grid of the modelled ejector, a 3D case

Numerical modelling has used the reconstruction schemes of the second degree and the k-ε standard turbulence model. Aufficiently good convergence has been obtained during the iteration process (mass residual values below 10^{-3}, velocity and energy residual values, k and ε at the level of $10^{-7}-10^{-8}$), which confirms the correct choice of grid for the existing gradients.

4. Results

Presented below is a comparison of the diffuser performance characteristics obtained in the numerical experiment with their reference experiment results. All designations and definitions of the given constituents are taken from the reference article [1], which describes the experiment.

Values presented in the charts:

- The Mach number in the diffuser inlet section:

$$M_1 = \frac{w_{1\mathrm{avg}}}{a_{1\mathrm{avg}}}, \tag{1}$$

where $w_{1\mathrm{avg}}$ – average velocity in the diffuser inlet section x_1, $a_{1\mathrm{avg}}$ – average speed of sound in the diffuser inlet section x_1.

- The pressure loss coefficient in the diffuser:

$$\delta_d = \frac{\bar{p}_{02}}{\bar{p}_{01}}, \tag{2}$$

where $\bar{p}_{01}$ – average total pressure in the diffuser inlet section x_1, $\bar{p}_{02}$ – average total pressure in the diffuser outlet section x_2.

- The relative total pressure drop in the diffuser:

$$\frac{\Delta p_0}{\bar{p}_{01}} = \frac{\bar{p}_{01} - \bar{p}_{02}}{\bar{p}_{01}}. \tag{3}$$

- The pressure rise (compression) coefficient:

$$\delta = \frac{\bar{p}_2}{\bar{p}_1}, \tag{4}$$

where $\bar{p}_1$ – average static pressure in the inlet section x_1, $\bar{p}_2$ – average static pressure in the outlet section x_2.

- Diffuser efficiency:

$$\eta_d = \frac{\bar{p}_2 - \bar{p}_1}{(\bar{p}_{01} - \bar{p}_1) - (\bar{p}_{02} - \bar{p}_2)}. \tag{5}$$

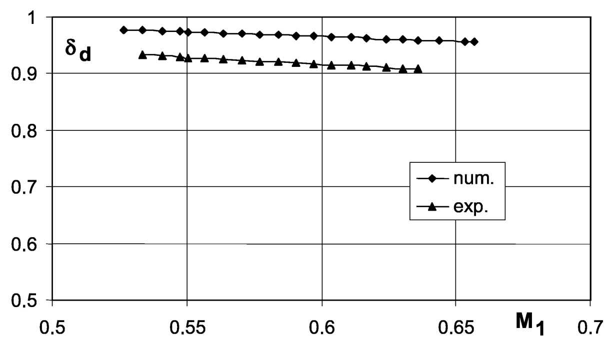

Figure 4. Variation of the pressure loss coefficient as a function of inlet velocity

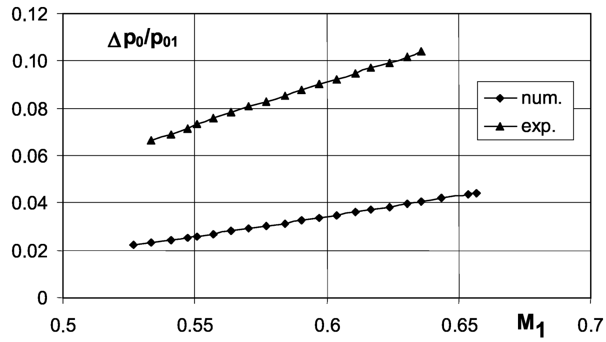

Figure 5. The relative total pressure drop in the diffuser as a function of inlet velocity

One of the results of the numerical simulation has been the performance characteristics $\Pi_s = f(\chi)$ (Figure 8). As the authors of reference article do not give such characteristics, it is difficult to determine its correctness. Inferring, however,

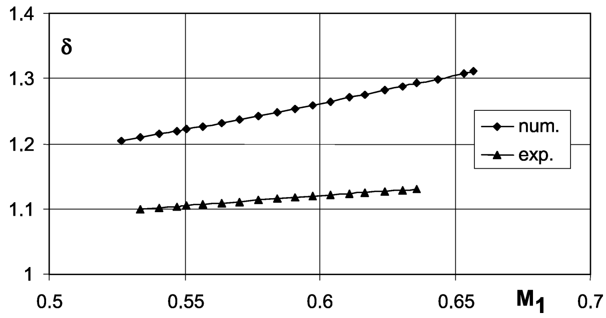

Figure 6. Variation of the pressure rise coefficient in the diffuser as a function of inlet velocity

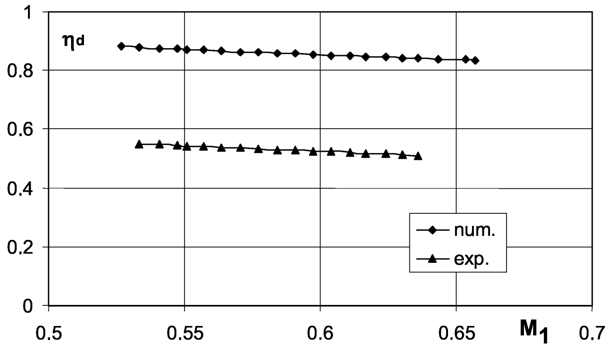

Figure 7. Diffuser efficiency as a function of inlet velocity

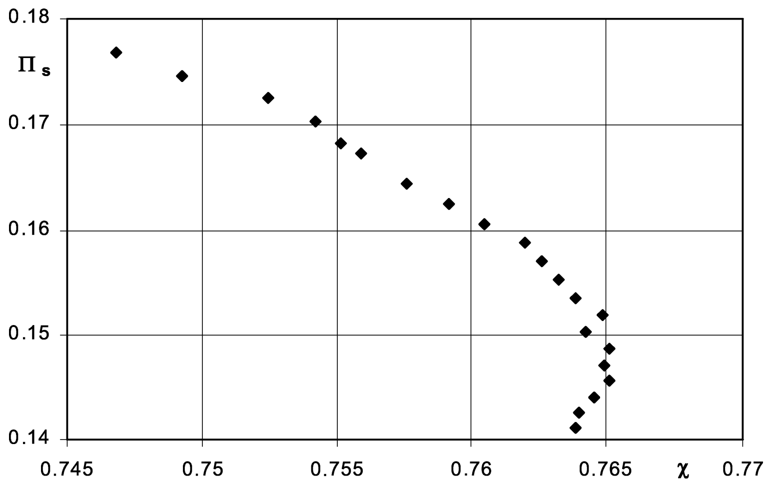

Figure 8. Ejector performance characteristics $\Pi_s = f(\chi)$ obtained by numerical simulation

from significant discrepancies in other characteristics presented above, it may be assumed that the performance characteristics obtained in the numerical simulation does not correspond to the actual characteristics either. It should be emphasized that in the literature available to the author there is no description of a similar numerical experiment, and presenting the performance characteristics of the ejector $\Pi_s = f(\chi)$ is a novelty, offering an insight into the potential of commercial numerical codes. Despite the absence of a quantitative comparison, one may note that generally the dimensionless ejector characteristics correspond to various experimental data presented in the literature [2].

5. Discussion

Figures 4 – 7 show a comparison of the numerical simulation results with the results presented in the reference article. The charts indicate that the numerical simulation results differ significantly from the actual results, artificially increasing the ejector parameters. This is particularly conspicuous in the case of efficiency – the diffuser of the virtual ejector has much higher efficiency than the real one.

The reasons of these discrepancies may be as follows:

- Incorrect choice of the absent ejector dimensions (the reference article [1] does not specify the dimension L_k) and greater than expected influence of the length and geometry of conduits feeding the media to and carrying them away from the ejector. However, it seems that a single *missed* dimension should not have such significant influence on the ejector performance characteristics and, regardless of the assumed geometrical dimensions, such unrealistic efficiency values (of more than 90%) should not be obtained. In the case of a correct numerical model and incorrect dimensions (in this case only one dimension), one should expect only insignificant deviation from the experimental characteristics. However, the described case proved otherwise, which supports the conclusion that geometrical differences could not have influenced such large discrepancies between the numerical and the experimental characteristics.
- Absence of adequately fitting *closures* and dissipative stages. Based on the obtained results, it may be concluded that the general numerical model is correct, but the closures relating to turbulence or other physical phenomena that play an important part in case of ejectors may be wrong. This hypothesis is confirmed by the fact that the general run of numerical characteristics conforms to the experiment; the characteristics are only shifted in relation to the experiment. This may mean that the numerical model does not include descriptions of some dissipative phenomena that exist in a real ejector: the *too good to be true* performance of the model ejector indicates so. Another reason may also be the fact that numerical modelling in the fluid mechanics has so far focused mainly on fluid-flow machines, such as pumps, turbines or reversible machines, and the prepared numerical models might be much less universal than they are commonly thought to be. Thus, in their present state, they might not be suitable for the modelling of ejector equipment.
- Absence of unsteady phenomena (bifurcations) in the area of jet outflow from the working nozzle to the mixing chamber. Laboratory experiments and many numerical-based works indicate the possibility of losing the stability of the jet when it flows out of any type of nozzle or canal into a free space, *e.g.* an ejector mixing chamber. In this case, there are two possibilities [3, 4]:
  - irregular bifurcations, characteristics of which is dependent on geometrical imperfections and on momentary imperfections caused by feeding and carrying away systems, the mixing chamber or the diffuser,
  - stable Hopf bifurcations (Figure 9).

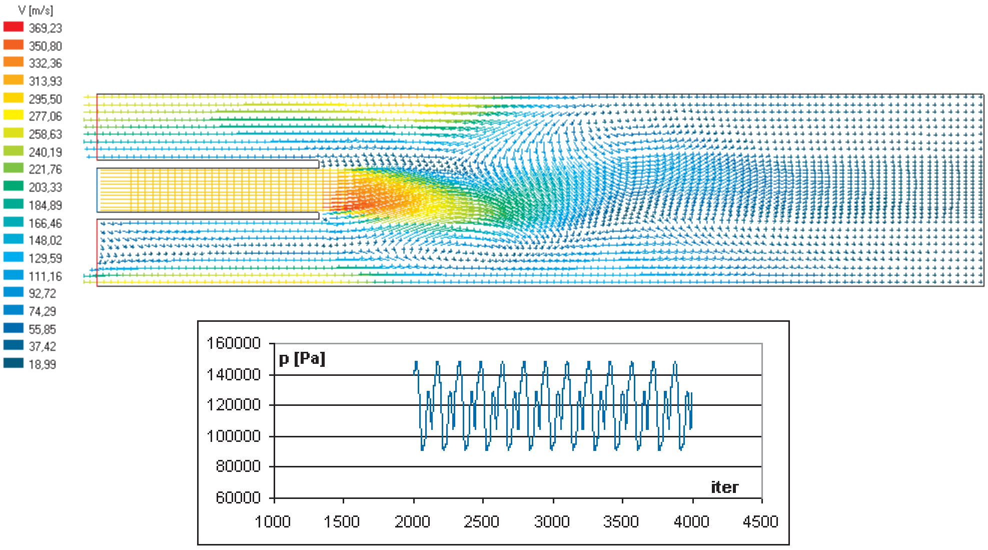

Figure 9. An example of flow with Hopf bifurcations (top); pressure–iterations diagram (bottom)

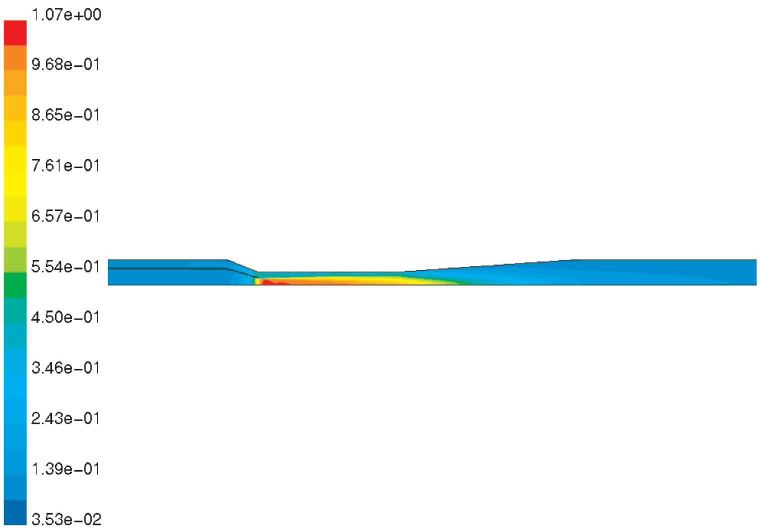

Figure 10. Mach number distribution for $p_r = 179\,500$

The examined ejector performance computer model has been characterized by total *rigidity* of the working fluid jet and no jet stability loss could be noticed (Figures 10 and 11), although a wide range of ejector working parameters has been examined (pressure at the working nozzle inlet).

- Absence of unsteady phenomena (bifurcations) in the boundary area between the working medium and the sucked medium [3–5]. The momentum exchange was generally taking place due to viscosity, and no whirl was observed on the boundary of the components (Figures 10 and 11).

It is however known that various whirls may occur during coaxial flow of two components [4, 6, 7]. This phenomenon is observed both during experimental and

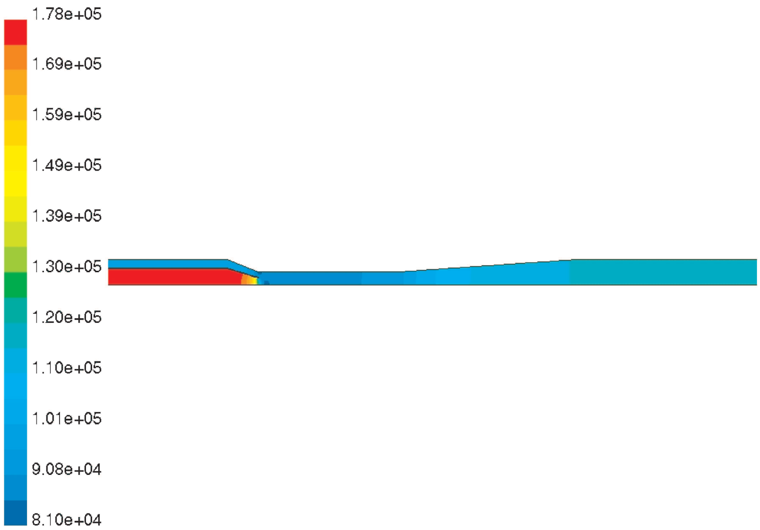

Figure 11. Static pressure distribution for $p_r = 179\,500$

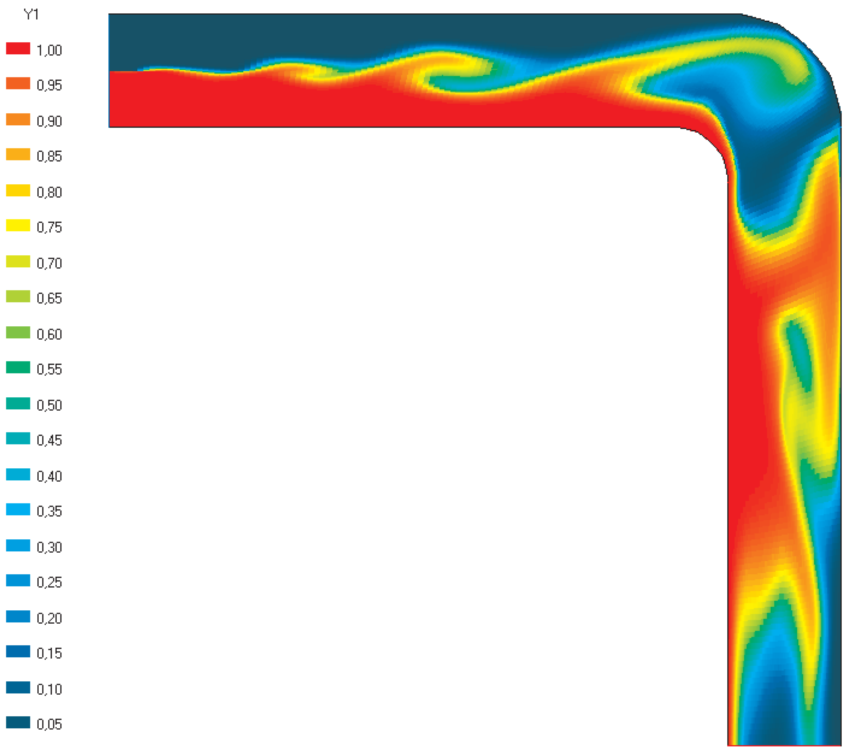

Figure 12. An example of whirls in the boundary zone during parallel flow of two components [4]

numerical research. An example of the latter is parallel flow of two gases in a canal with elbow, in which – due to an obstacle (elbow) – the whirls change over time (Figure 12). A particular feature of this flow is that the whirls are created already before the obstacle, even though the assumed model has been non-viscous and without turbulence.

6. Conclusions

The conducted research indicates the necessity of further development of numerical models applicable to ejector equipment. The comparison has proven that using the FLUENT 5.5 solver and standard calculation techniques does not yield satisfactory results.

Therefore, a full simulation of ejector equipment performance suitable for practical purposes is presently impossible. This is due to still inadequate and very specialized possibilities of numerical codes. The abundance of physical phenomena occurring in the flows makes construction of universal models substantially difficult; also ejector equipment requires a dedicated approach and preparing appropriate numerical techniques.

It should be emphasized that the material presented in the paper does not exhaust the possibilities of modern numerical codes. It also indicates the directions of further research in the following areas:

- the influence of viscosity on the *rigidity* of a working fluid jet and on the flow behaviour on the boundary of two components;
- the influence of various turbulence models on numerical simulation results (also the models added to latest versions of the FLUENT package);
- directions of development of mathematical models (in relation to ejectors) and their possible implementation in commercial numerical codes.

Acknowledgements

This paper was carried out within a framework of research project no. 5 T07B 036 22.

References

[1] Werszko D and Goliszek A 1984 *Works of the Institute of Fluid-Flow Machinery*, Polish Academy of Sciences, Gdansk, **87** 41 (in Polish)
[2] Golinski J and Troskolański A 1979 *Ejectors*, WNT, Warsaw (in Polish)
[3] Badur J and Sobieski W 2001 *Research of the Chair of Mechanics and Bases of Machinery Constrution*, Olsztyn, Poland, pp. 63–98 (in Polish)
[4] Sobieski W 2002 *Mixing and Cavitation Phenomena Modelling in Gas-Fluid Ejectors*, PhD Thesis, Institute of Fluid-Flow Machinery, Polish Academy of Sciences, Gdansk, Poland (in Polish)
[5] Badur J and Sobieski W 2000 *Fluid Dynamics 2000*, Prague, Czech Republic, pp. 3–4
[6] Kudryński A W, Rusanow A W and Jerszow S W 1997 *Proc. Int. Sc. Conf. "Mathematical and Physical Modelling Methods for Development of Turbines"*, Kharkov, no pagination (in Russian)
[7] Roshko A 1976 *AIAA Journal* **14** (10) 1349