International Journal of Pure and Applied Mathematics
Volume 118 No. 20 2018, 27-31
ISSN: 1314-3395 (on-line version)
url: http://www.ijpam.eu
Special Issue

ijpam.eu

Numerical investigation of steam ejector characteristics for petroleum and petrochemicalsheating before transportation

Dmitry Pashchenko[1],*Ilya* Naplekov[1], and *Ivan*Zheleznikov[1,*]

[1]Samara State Technical University, 443100Samara, Russia, 244 Molodogvardeyskaya str.

Abstract. Presented work is focused on creating a computational model of steam ejector. The above mentioned steam ejector is suitable for petroleum heating, petrochemicals transportation and other different key functions of oil and gas industry. The commercial software ANSYS Fluent was used. The procedure of achieving of the objective includes an algorithm of adaptation of the computational grid, choosing the turbulence model, choosing the numerical setup for ANSYS Fluent solver, and estimation of the results. Within the calculations done, the validation of the model based on the experimental data has been conducted. It was concluded then, that the obtained by CFD-modelling results are corresponding well with the experimental data and deviation of less than 2-4% had place. The results are provided in the form of lines of flow, edges, and diagrams. Design conditions for the ejector for the established parameters were set. This paper includes description of the recommended numerical setup for solver for ANSYS Fluent for engineering calculations of steam ejectors.

1 Introduction

Steam ejectors have been widely used in almost all industries due to their special feature - the ability to increase the pressure of the gas flow without the direct cost of mechanical work with an exceptional simplicity of structures. Russian scientists such as Sokolov E., Zinger N., Berman L., Efimochkin G., and others were engaged in the generalization, systematization, and development of the principles of operation of jet devices. [1] .In recent decades, the principles of operation of several thousand different designs of jet devices have been proposed for consideration and have been studied [2] .In the oil and chemical industry, heat power engineering, steam ejectors have found wide application, the main purpose of which is the mixing of high and low-pressure steam flows, and production of steam at specified pressure and temperature parameters [3] .The variety of designs and methods for calculating these devices makes constant searches for new engineering solutions in the tasks of research, optimization, and production relevant.

In recent years, in engineering practice, more and more attention has been paid to numerical methods of computer modeling of various physical and chemical processes - CFD-modeling (Computational Fluid Dynamics) [4] .The main advantages of this method of scientific search are the possibility of obtaining a visual representation of the nature of the processes in the object under study and the possibility of studying its various designs without creating expensive experimental installations. And also the possibility of optimizing the principles of working with the purpose of obtaining the set parameters of the device operation, etc. According to NASA experts, by 2030 about 85% of the computing power of all supercomputers on Earth will be used to solve problems of computational fluid dynamics [5] .For effective application of CFD modeling one can use number of commercial programs, such as ANSYS, Flow Vision, Comsol Multiphysics, etc., and open source programs such as Open FOAM, Salome, Code Saturn, and others.

The purpose of this work is to develop a computational model of the steam ejector using software product ANSYS Fluent, its verification using experimental data, as well as carrying out a research of the dependence of the main parameters of its operating conditions. Modern software and computational tools make it possible to produce a wide range of studies of various devices without creating physical models for these objects. Whether full or partial replacement of the

[*]Corresponding author: zheleznikov_is@icloud.com

physical experiment for the majority of jet devices with numerical computer simulation is possible, will depend on a comprehensive study of various turbulence modes and continuous improvement of CFD modeling algorithms.

2 Methods

The object of investigation is a steam ejector, which geometric characteristics are shown in Fig.1. In this ejector, high-pressure steam flow (Input 1) and low-pressure steam flow (Input 2) are mixed, producing steam (Output) with the specified pressure parameters.

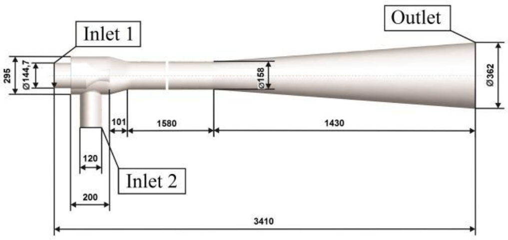

Fig. 1. Geometric characteristics of the steam jet ejector.

The task of CFD modeling is solved both in two-dimensional (to determine the optimal settings of the solver Fluent) and in the three-dimensional problem definition. Geometry is constructed in the Autodesk AutoCAD automated design system . The calculated geometry completely repeats the geometric characteristics of the real ejector.

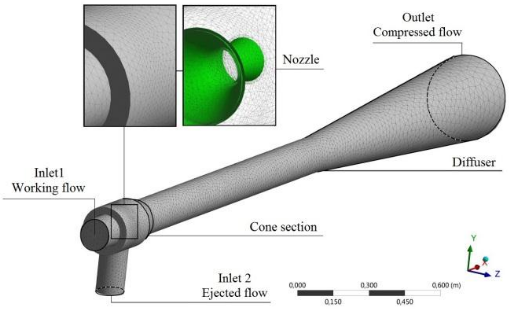

Fig. 2. Computational mesh

To perform a numerical experiment computational geometry has been meshed. Increasing the accuracy of the solution is accomplished by adapting the mesh in the Meshing module built in the ANSYS .In order to increase the number of final volumes, the parameter responsible for the density of the calculated cells (Relevance) was set at 95.The total number of grid elements is 492424, while the minimum value of the element's edge is 1.27 mm, the maximum value is 250 mm.General view of the computational mesh is shown in Fig. 2.

In the Setup of the Fluent Solver, the mathematical description of the flow process is expressed by a system of differential equations consisting of continuity equations, the law of conservation of momentum, and the law of conservation of energy, generally described in Ansys Fluent 14.0: Theory Guide [6].

To close the Navier-Stokes equations [7], which describe the motions of the steam in the ejector, we use the k-ε Realizable model of turbulence with improved wall functions that take into account linear and logarithmic sections in the velocity distribution in the near-wall layer. To simplify the calculation due to high steam velocities, a zero gradient of the heat flux through the wall is established in the Fluent settings - there are no heat losses through the wall [8-9].

Due to a large number of mesh elements, hybrid initialization is carried out in 10 iterations. For the same reason, the number of iterations for the time step is set to 30. An increase in the number of iterations with a high quality of the mesh greatly complicates the computations, without giving high accuracy. The time step is set to 0.001 s, their number is 100 for the two-dimensional problem and 10 for the three-dimensional one.Such a number of time steps makes it possible to obtain an established picture of the outflow throughout the entire space of the gas flow.

3Results and discussion

Verification of the model is carried out in two stages. At the first stage, the settings of theFluent solver were determined therefore they allowed to gain the allocation of the velocity and pressure contours in the ejector, most accurately corresponding to the actual processes.For this purpose, the high-pressure steam outflow process into a free volume with atmospheric pressure has been investigated. Fig. 3 shows the results of a numerical study of the process of gas outflow into the free volume (a) and comparison of simulation results with experimental data (b).

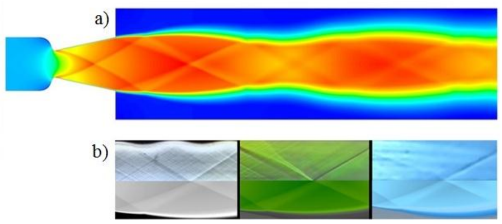

Fig. 3. a) outflow into the free space; b) in the upper half-plane there are results of the experiment, in the lower half-plane there are models with different color filter

Fig. 3(b) shows the surprint of the velocity contour obtained by experiment (upper part) with the results of calculations (bottom part). A diamond shock wave is observed for the velocity contour, characteristic of all outflow processes with a Mach number more than one (M > 1).The solution of the problem of modeling the flow into free volume is an important stage of verification and validation of the design model.In case of coincidence of the velocity contours obtained in the numerical and computational experiment, it can be asserted that the selected solver settings and the turbulence model constants describe the physical process as accurately as possible. Fig. 3b shows a full match of numerical and experimental velocity contours, the change in fill color is performed in the CFD-Post module.

The second stage of verification of the developed model was carried out in the three-dimensional formulation of the problem for the computational domain shown in Fig.1. The total error of the results for the pressure was 2.2%, for the temperature 1.6%. For other design modes, the errors also did not exceed 2-3% for both, pressure and temperature.

Table 1. Results of model verification.

	Massflow, kg / s	Pressure, MPa	Temperature, K
Workflow (Inlet 1)	17,34	1,317	487,00
Injectedstre am (Inlet 2)	4,886	0,147	385,00
Compressed flow (Outlet)	22,22	0,405	416,00
	22,39	0,414	416,67

The velocity distribution in the steam ejector is shown in Fig. 4 for the initial data given in Table 1. A high-pressure steam stream (1.317 MPa) outflowing from the nozzle, injects a low-pressure steam stream (0.147 MPa), resulting in vapor at a pressure of 0.414 MPa. Three-dimensional visualization of the trajectories of the steam current lines, and pressure, temperature and velocity contours is shown in Fig.5.

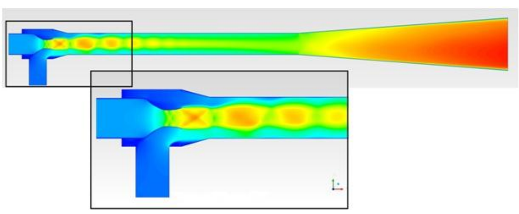

Fig. 4. Velocity profile in steam ejector

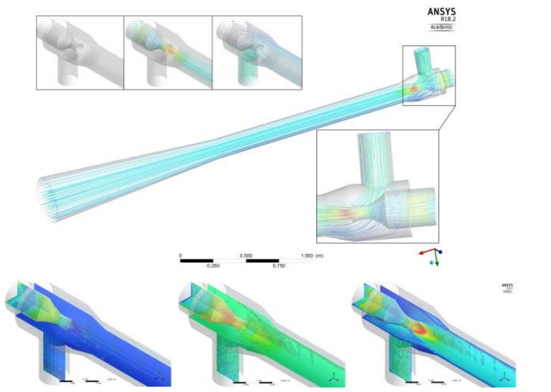

Fig. 5. Three-dimensional visualization of results. A) Trajectories of steam flow lines in the ejector. B) Profiles: From left to right - pressure, temperature, velocity.

4 Conclusion

The examples and results shown above illustrate that modern numerical methods for solving the problems of computational fluid dynamics make it possible to reproduce as completely as possible the physical experiment with minimal errors not exceeding several percent. Thus, the need to conduct time-consuming and financial wasteful physical experiments related to the determination of operating conditions for steam ejectors for heating oil products of various compositions and at different temperatures is substantially reduced. A numerical computer experiment can come to the fore. However, without reliable results of physical experiments and their comparison with the results of modeling, it is impossible to speak about the adequacy of the developed models.

For engineering calculations of jet devices, in particular, steam ejectors, it is recommended to use the following settings of the Fluent solver in the software product ANSYS, which allows to describe as accurately as possible the physical processes taking place in these devices:

- the type of solver is pressure based;
- connection of the energy conservation equation - energy equation "On";
- operating fluid - steam;
- density calculation - ideal gas (ideal gas);
- atmospheric pressure environment (operating conditions);
- setting the boundary conditions through the mass flow (mass flow inlet);
- for all state parameters of a second order equation (second order upwind);
- Implicit form of the first-order equation for the non-stationary calculation (first order implicit);
- hybrid initialization scheme (hybrid initialization).

References

1. T.Sriveerakul, S. Aphornratana, K. Chunnanond, International Journal of Thermal Sciences,**46,**8 (2007)
2. Y.M. Chen, C.Y. Experimental Thermal and Fluid Science,**15,** 4 (1997)
3. X. Ma, C.Y. Sun, S.A. Omer, S.B. Riffat, Applied Thermal Engineering,**30,** 11-12 (2010)
4. D.I. Pashchenko, MATEC Web of Conferences,**145** (2018)
5. J. Slotnick, A. Khodadoust, J. Alonso, D. Darmofal, W. Gropp, CFD vision 2030 study: a path to revolutionary computational aerosciences, (2014)
6. J.Slotnick, CFD vision 2030 study: a path to revolutionary computational aerosciences, 2014
7. Ansys Fluent 14.0: Theory Guide (2011)
8. D. Paschenko,International Journal of Hydrogen Energy,**42,** 49(2017)
9. D. Pashchenko, Energy,**143** (2018)