ANNALS of Faculty Engineering Hunedoara
– International Journal of Engineering

Tome XII [2014] – Fascicule 2 [May]
ISSN: 1584-2665 [print]; ISSN: 1584-2673 [online]
a free-access multidisciplinary publication
of the Faculty of Engineering Hunedoara

[1.] Juraj KRAJANEC

POSSIBILITIES OF FLOW SIMULATION IN LAVAL NOZZLE

[1.] Faculty of Mechanical Engineering, STU in Bratislava, Námestie slobody 17, 812 31 Bratislava, SLOVAKIA

Abstract: This paper was inspired by the field of ejector design and deals with the flow in the Laval's nozzle. The problems of achieving of a low pressure in a vacuum ring pump are described in the introduction to the paper. Then the machine is precisely classified into the machine category and work principle of the ejector is described, as well. There is explained also the advantage of using the preliminary ejector within a vacuum ring pump. The paper deals with the principle of ejector operation and its particular design parts. In the following part of the paper is a description of a meshing in the software GAMBIT and setting of the boundary conditions for CFD simulation in the software ANSYS, which is used for CFD analysis.
Keywords: Laval's nozzle, flow simulation, work principle

1. INTRODUCTION

Devices to create vacuum or underpressure are used for extraction of gases and vapors from different environments and facilities. These devices include liquid ring vacuum pump. Performance data of these vacuum pumps are limited by saturation pressure. The suction pressure cannot fall below the vapor saturation pressure due to pump cavitation and its subsequent destruction. To reduce the end vacuum through vapor saturation pressure and to keep original pressure on the suction side of the pump upstream ejectors are used.

Ejectors are fluid machines and contain neither moving parts nor seals, and therefore they are simple and nearly maintenance free. Devices to produce vacuum are propelled by surrounding air to keep running costs constant. Single stage ejector and serial connection of multiple ejectors are proposed according to required vacuum level. For proper operation of the ejector and the best impact on reducing the vacuum it is necessary to understand and properly analyze the flow in the ejector. To analyze the flow inside the ejector we used CFD (Computational Fluid Dynamics), to determine the pressure field, velocity field, turbulence and many other important flow factors.

2. FLOW AND MIXING PROCESS IN THE EJECTOR

Generally ejector consists of primary nozzle, secondary nozzle, mixing chamber and diffuser. Confusors are usually placed before mixing chamber in the case of equal-pressure ejector. Upstream ejectors are usually supersonic. Primary nozzle frequently shaped as Laval nozzle. The Surrounding atmospheric air (101325 Pa) works as a flow transmission medium Narrowest place of the nozzle is designed to reach the speed of sound M=1[-] at that point. Laval nozzle of the ejector has the major influence on ejectors performance. Ejectors can operate in four different modes:

- Operating mode 1 (Figure 1): The pressure drop in primary nozzle is above critical value, but it is insufficient to expand to the end of the nozzle, and the flow within the nozzle accompanied by a strong shock wave is separated from the walls and is narrowing down. The total secondary flow is less than the maximum.

ISSN: 1584-2665 [print]; ISSN: 1584-2673 [online]

- Operating mode 2 (Figure 2): The primary flow expands into the end of nozzle and after leaving the nozzle leading to a narrower stream dependent on the value of the secondary pressure p1" or is accompanied by an sloping shock wave or pressure difference in the outlet section of the primary current passes through elementary expansion wave with the growing shape.

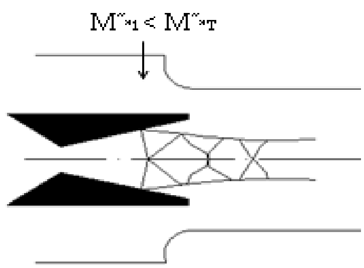

Figure 1: Operating mode 1

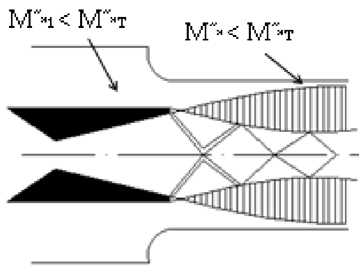

Figure 2: Operating mode 2

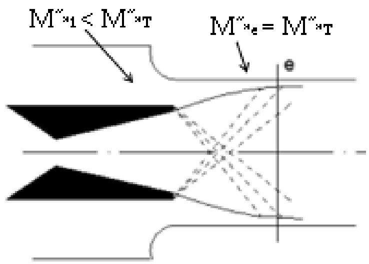

Figure 3: Operating mode 3

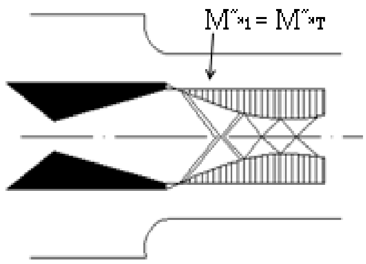

Figure 4: Operating mode 4

- Operating mode 3 (Figure 3): primary flow expands further with expanding shape after leaving the nozzle and gives some of its energy to secondary stream. Secondary flow accelerates in the nozzle, with reduced diameter created by the wall of the mixing chamber and the border of the primary stream to the extent that section labeled "e" is reaching critical Mach number.
- Operating mode 4 (Figure 4): Secondary flow is so strong that exit Mach number at the end of the nozzle reaches critical value and the secondary flow reaches its maximum.

Ejectors are designed for operating mode 3 [1], [2], [3] when mass flow through the primary nozzle is highest m_1 = max [kg / s] and the mass flow through the secondary nozzle is m_2 = 0 [kg / s]. Basic preconditions for calculation of the characteristics of an ejector [2]:

a) Primary and secondary gas can be considered as an ideal gas and have the same isentropic exponent and gas constant.
b) The ejector is completely thermally insulated and nozzles are mutually insulated.
c) Both streams undergo polytrophic change of state till the separation section.
d) The flow is one-dimensional.
e) Mixing chamber is long enough to mix streams thoroughly.
f) The temperature of the primary and secondary gas in the entity can also be different.

Mach number:

$$M = \frac{w}{a} \quad [-] \tag{1}$$

Critical pressure:

$$p_k = p_o \left(\frac{2}{\chi + 1}\right)^{\frac{\chi}{\chi-1}} \quad [Pa] \tag{2}$$

Specific volume at the critical point:

$$v_k = v_0 \left(\frac{p_0}{p_k}\right)^{\frac{1}{\chi}} \quad [m^3/kg] \tag{3}$$

Critical velocity:

$$w = \sqrt{\chi p_k v_k} \quad [m/s] \tag{4}$$

where: p_0 – pressure at nozzle entry [Pa], v_0 – specific mass at nozzle entry [m^3/kg]

Flow in Laval nozzle is simulated into open space, resp. tank. The computational domain consists of the interior of the Laval nozzle and tank. The tank has to be large enough to allow the flow to develop. Flow in Laval nozzle is generally three-dimensional; nevertheless the two-dimensional flow field was assumed, to simplify meshing, calculations and analysis, and consequently reduce calculation time and reduce hardware requirements for computing.

CFD simulations work on the principle of dividing the computational domain into finite number of volumes. The governing equitations are implemented in CFD model, in order to simulate flow phenomena. CFD model assumes the compressible flow of the medium. Meshing was performed in GAMBIT software. Optimum size of grid of the nozzle is 0.4 mm. Higher mesh density can cause the divergence of the solution. In general, the structured mesh scheme is preferred. If it is not possible to utilize the structured mesh scheme, the unstructured quadrilateral PAVE mesh scheme might be implemented instead. The boundary zones – axis, wall, inflow and outflow have to be set before exporting the mesh from the GAMBIT software.

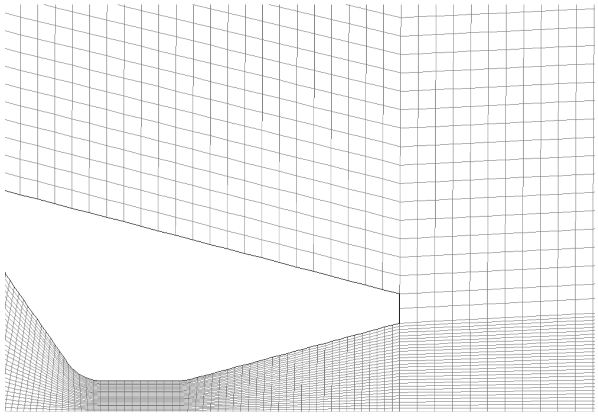

Figure 5: Mesh in Laval nozzle and tank

3. CFD SIMULATION

From GAMBIT software we export computational mesh to ANSYS. Then it is necessary to specify the basic parameters of the CFD model as follows: scale of the model, axially symmetric model, the working medium, turbulence model and boundary conditions. In ANSYS solver we define task as a time-settled "steady" and type "Density - Based". As a working medium, we set the ideal gas and turbulent model k-omega SST. Inlet pressure of the vacuum pump is the same as the pressure downstream of the ejector. Ejector is designed for operating mode 3, mass flow rate is defined for the inlet primary nozzle and flow rate in secondary nozzle is zero. According to characteristics of the vacuum pump we change only the inlet mass flow rate and outlet pressure for diffuser. After converged calculation, solution is obtained and the mesh adaptation might be performed according to specified value of some quantity. It can be adapted according to the network increase some physical quantity. In areas with large gradient values, the network solver refines the network model to refine results. Especially in the areas with high gradient values the mesh refinement is performed in order to achieve more accurate results. Accuracy of the nozzle design can be verified with ANSYS, by setting the proposed mass flow rate for the nozzle and by setting display of absolute pressure, which is 101 [kPa]. Comparison of Figure 1 and Figure 6 clarifies that operating flow conditions in Laval nozzle / ejector can be observed in results of CFD simulation. Similar results were obtained in [4], [5] which have dealt with an ejector for the cooling system and CFD simulation. In [4], [5] to study the ejector flow not only the Laval nozzle flow authors used different boundary conditions and different turbulent model.

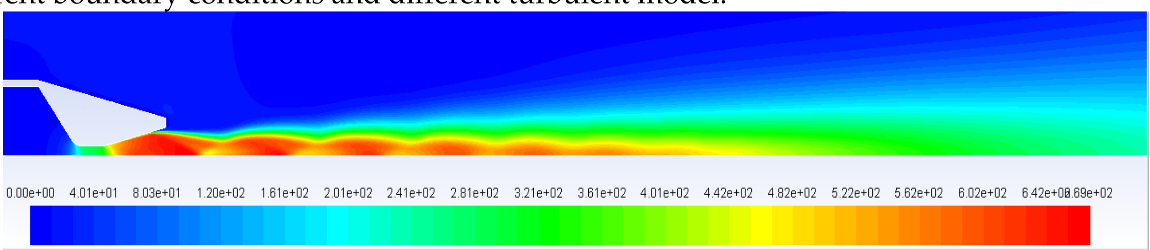

Figure 6: Velocity flow of the working substance in the Laval nozzle into open space

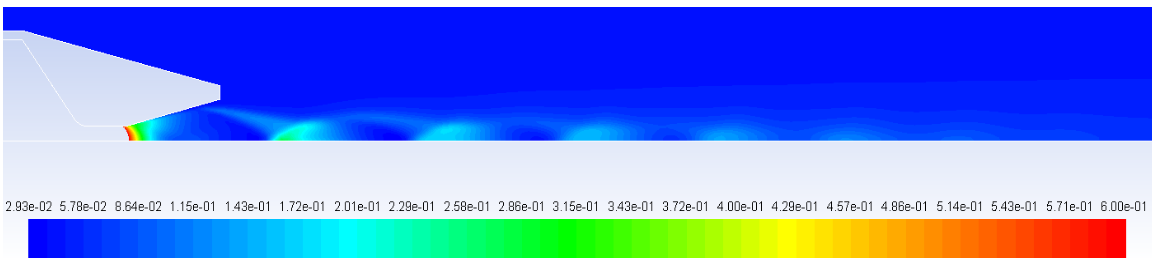

Figure 7: The local density change in flow

ISSN: 1584-2665 [print]; ISSN: 1584-2673 [online]

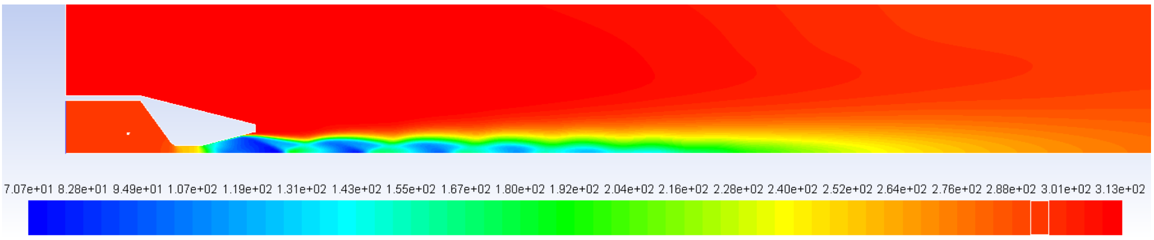

Figure 8: Temperature change in the flow

4. CONCLUSION

As shown in Figure 6 to 8, CFD simulation is capable to capture pressure wave, velocity of the stream and density change field accurately. In comparison to Figure 1 to 4, we can state that the proposed nozzle, respectively generator is going to work in the ideal operating mode. With results of CFD simulation, we are able to verify that proposed Laval nozzle is able to achieve required and calculated parameters. More importantly, by using CFD simulations we are able to record and eliminate the formation of local vortices and other negative impacts of the suboptimal proposed nozzle.

REFERENCES

[1.] Szabó Szilárd, Výpočetní postup pro konstruování injektoru plynu k vakuovému čerpadlu s vodním prstencem a pro stanoveni charakteristik jejich společného provozu, 1982, Miskolc,

[2.] Olšiak Róbert, Kvapalinokružné vývevy s predradeným ejektorom, 2000, STU Bratislava, ISBN 80-227-1402-X

[3.] Löser Helmut, Výzkumy na jednostupňových nadzvukových ejektorech s válcovou směšovací komorou, 1965, Damrstadt

[4.] K. Pianthong, W. Seehanam, M. Behnia, T. Sriveerakul, S. Aphornratana, Investigation and improvement of ejector refrigeration systemusing computational fluid dynamics technique, 2007, www.sciencedirect.com, Energy Conversion and Management 48 (2007) 2556–2564

[5.] Amel Hemidi, François Henry, Sébastien Leclaire, Jean-Marie Seynhaeve, Yann Bartosiewicz, CFD analysis of a supersonic air ejector. Part II: Relation between global operation and local flow features, www.sciencedirect.com, Applied Thermal Engineering 29 (2009) 2990–2998

ANNALS of Faculty Engineering Hunedoara – International Journal of Engineering