UDC 532.5

Ye.O. Shkvar, Dr. of Sci.; Shi-ju E, PhD; A.S. Kryzhanovskyi, PhD
(College of Engineering at Zhejiang Normal University, P.R. China)

Numerical simulation of a turbulent axisymmetric jet with internal local swirl

A new simple and efficient approach of modelling the hot turbulent axisymmetric jet inside the nozzle and in its outlet neighbourhood with the aim of angular homogeneity improvement is proposed, verified and analysed on the basis of both elaborated theoretical model and the results of experimental measurements. The level of achieved agreement is quite appropriate for engineering applications.

Introduction.

There are a lot of various engineering applications of jet flows and as usual their mode of development is turbulent. One of very efficient methods of forming and control of jet flow with required properties is swirling. Jet is one of quite simple, canonical and well-studied flow types, but jet with swirling can be characterized by complexity of its structure with dualism of its properties and numerical solutions depending on a swirling criterion *Sw* [1, 2]. As a result, this promising method of flow parameters control requires careful adjustment to the estimated area of jet development, therefore the problem of reliable optimizing the swirling jet generation, based on modern methods of numerical modelling is both important and perspective.

Problem statement.

Let's consider a hot jet, generating by some typical heating machine, that passes through the nozzle head 1 with external deflector 2. Let this nozzle head be located directly behind the ceramic heater 3 (fig. 1) and let the maximum jet speed be about 25 m/s and temperature in the outlet of the heater 4 is – 600^{o}C.

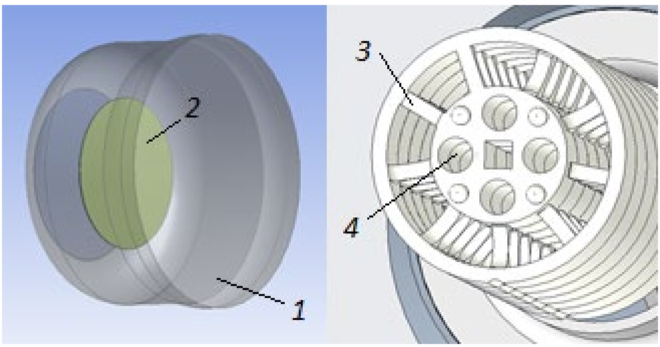

Fig. 1. Principal elements of a nozzle

The basic geometrical sizes of the nozzle elements are: external diameter of the head 1 is 35mm, outlet diameter is 23, rounding radius of 1 is 6mm, height (distance from outlet section of ceramic heater to the head outlet) – 18mm, distance between nozzle head 1 outlet and external deflector 2 is 7mm and the diameter of the external deflector 2 is 20mm.

The goal of this research problem is to reduce as much as possible the negative influence of technically caused structural inhomogeneities (heater baffles 3, several technological holes 4 in its core) to ensure the angular uniformity of flow parameters in the jet and develop the corresponding numerical model of this kind of turbulent swirl axisymmetric jet flow.

Aerodynamic design of the proposed technical solution.

To obtain the most efficient technical solution for achieving angular uniformity, several different approaches have been tested experimentally. The most effective method of them is to create a local flow swirl near the core of the heater 3. For this purpose, the simplest solution was proposed – to install additional internal deflector 5 (fig. 2) at the end of the heater core in order to redistribute the energy of the flow passing through the internal technological holes into the energy of the swirling flow behind the heater core. In order to generate swirl flow an array of small inclined blades 6 was organized at the edge of the internal deflector.

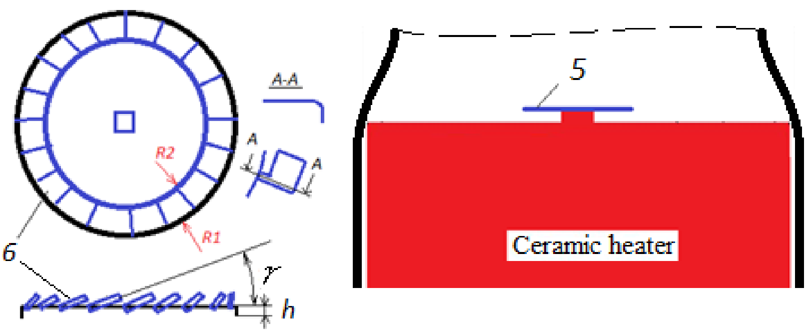

Fig. 2. Internal deflector of the heater core

This idea has demonstrated its fruitfulness in experiments, but for fine tuning the geometry of the deflector and blades (values R1, R2, h, angle), a mathematical model was developed.

Governing equations.

To simulate the jet flow from the heat gun nozzle, the following system of Reynolds-averaged Navier–Stokes (RANS) governing equations has been solved under the assumption of stationary incompressible (but temperature dependent) turbulent flow that completely corresponds to the formulated above working conditions:

$$
\begin{cases}
\nabla\rho\bar{V} = 0; \\
(\bar{V}\cdot\nabla)\bar{V} = -\nabla p/\rho + \nabla\cdot\left(\nu\nabla\bar{V} + \sigma_{t\,ij}\right); \\
(\bar{V}\cdot\nabla)T = \nabla\cdot(\lambda\nabla T + q_t); \\
p = \rho RT,
\end{cases}
\qquad (1)
$$

where $\overline{\sigma}_{tij} = -\overline{u_i' u_j'}$ are the additional Reynolds stresses that are the result of turbulent exchange dissipative mechanism, p is the pressure, T is the temperature, $\bar{V}$ – flow velocity, R=287 J/(kg K), $q_t = \lambda_t \nabla T$. The order of accuracy of the finite-volume discretization was chosen as second, and due to slowly convergent iteration process, the residuals for all computational variables were taken as $\varepsilon = 5\cdot 10^{-5}$.

Computational domain and mesh.

The computational domain was chosen according to the dominant direction of flow development. Then it was covered by structured mesh, having very small cells and detailed structure in the neighborhood of the wall surfaces, nozzle and input sections (fig. 3). This mesh is structured and well-adjusted to free jet-flow.

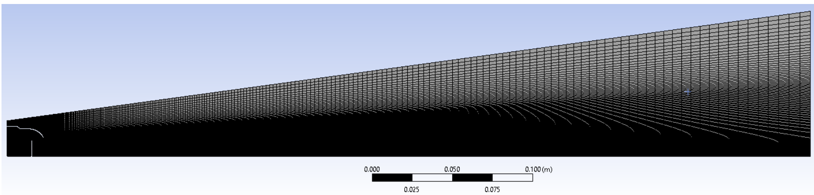

Fig. 3. Geometric model of the nozzle and computational mesh

Boundary conditions

The boundary conditions were established with taking into account the motion of ground surface relative to train body according to the ANSYS Fluent formalism as follows: All faces of surfaces of a nozzle and deflectors — Velocity Magnitude 0 m/s (wall); External lateral (top) and outlet (right) surface of domain — Gauge Pressure 0 Pa (pressure-outlet); Axis of flow – “Axis” boundary condition; Flow input (left) face surface of domain – Inlet (Velocity Magnitude determines by the profile, associated with the geometry of outlet of ceramic heater for the flux 500 l/min). In particular, the input velocity was determined as nonuniform in radial direction, especially, in the ceramic core region (0≤r≤6mm) velocity was taken as 0 and in the region with small holes (7mm≤r≤9mm) velocity was taken as averaged value for its value in the small holes.

Turbulence model.

According to the Boussinesq approach, the Reynolds stresses can be directly connected with the strain rate tensor components similarly to the laminar case, but by the use of additional viscosity (so-called eddy- or turbulent viscosity ν_t) that reflects the dissipative mechanism of the turbulent vertical system:

$$\overline{\sigma}_{\mathrm{t}ij} = -\overline{u_i' u_j'} = \nu_{\mathrm{t}} \overline{S}_{ij}, \quad \nu_{\mathrm{t}} = \mu_{\mathrm{t}} / \rho. \tag{2}$$

On the basis of RANS approach, the turbulent viscosity ν_{t} must be modeled with the use of additional semi-empirical model of turbulence. Within the framework of ANSYS software, the Spalart–Allmaras turbulence model was chosen as the most appropriate for this kind of a problem (1, 2).

Numerical model verification.

The velocity and temperature profiles of the turbulent jet flow from the heat gun nozzle have been tested experimentally and predicted numerically (fig. 4). As it follows from the comparison, the numerical predictions, obtained on the basis of the developed numerical model, correlate well with the experimental data for the analyzed jet flow.

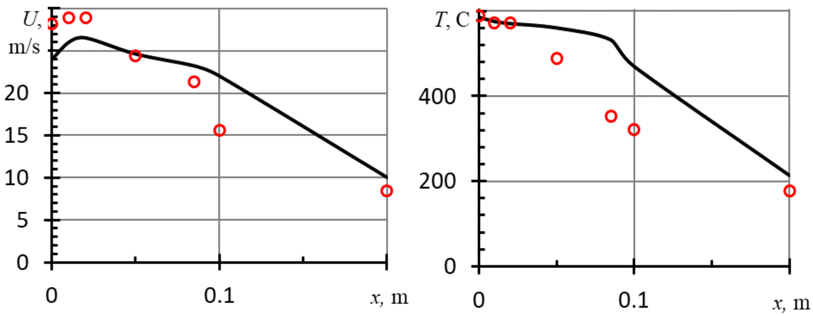

Fig. 4. Geometric model of the nozzle and computational mesh

Results of numerical simulation.

The results of numerical simulation of flow with internal impeller are shown below (fig. 5-7). All these illustrations demonstrate the significant intensification of vortical flow intensity and mixing in the region between two impellers as a result of installation of the internal impeller.

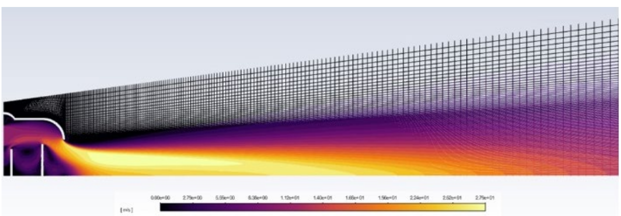

Fig. 5. Axial velocity distribution in jet flow through the nozzle head with internal deflector

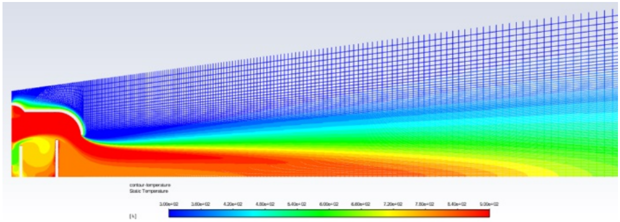

Fig. 6. Temperature distribution in jet flow through
the nozzle head with internal deflector

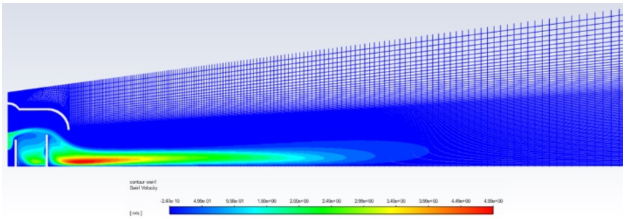

Fig. 7. Swirl velocity distribution in jet flow through
the nozzle head with internal deflector

Conclusions

1. The analyzed flow is characterized by strong radial and significant angular nonuniformities.
2. The theoretical model of flow development has been worked out and successfully verified by the corresponding experimental data (Fig. 4). This model allowed to find the optimized superposition of the nozzle head parameters.
3. The internal deflector with special design has been elaborated for covering the ceramic heater core with small holes. The results of experimental testing and numerical simulation proved that installing such a deflector allowed to substantially improve the flow parameters uniformity and increase the peripheral velocity.
4. The generated local internal swirling flow in the nozzle head strongly changes the further jet structure and parameters by creating the intensive swirling core, whose further development and interaction with the rest of flow will be the subject of further analysis and efforts application.

References

1. Shtern, V., Hussain, F. and Herrada, M. (2000) New features of swirling jets. Physics of Fluids, 12, 2868-2877. doi:10.1063/1.1313547
2. Yang, Y., Zou, D., Sun, W. *et al.* A swirling jet from a nozzle with tangential inlets and its characteristics in breaking-up rocks. *Pet. Sci.* **9,** 53–58 (2012). doi:10.1007/s12182-012-0182-0