SONIC UNDEREXPANDED JET IMPINGING ON THE PAIR OPEN TUBE – INNER CYLINDER

V. I. Pinchukov

Abstract- Interactions of sonic underexpanded jets with cylindrical bodies, placed in open tubes, are studied. Two-dimensional Reynolds averaged Navier-Stokes equations added by an algebraic turbulence viscosity based on the Karman length scale formulae are solved by an implicit third order Runge-Kutta scheme. Self-oscillatory regimes are found in CFD studies.

Keywords- Self-Oscillatory Flows, Open Tubes, Inner Bodies, Reynolds-Averaged Navier-Stokes Equations, High Resolution Methods, Runge-Kutta Schemes

I. INTRODUCTION

Recent paper is devoted to a search for new self-oscillatory flows. This search is started in [1] on a base of the hypothesis, that self-oscillations appear as a result of resonance interactions of “active” elements of flows, namely, elements, which amplify disturbances. Supposition is used about two types of “active” elements – contact discontinuities and intersection points of shocks with shocks or shocks with contact discontinuities. Possibility of the disturbances amplification by contact discontinuities is a result of the Kelvin-Gelman-Golstee instability and is accepted. Inclusion of intersection points to a list of amplifiers is proposed in [1] as a hypothesis, which is checked by results of a search for new unsteady flows.

List of self-oscillatory flows, resulted from resonance interactions of “active” elements includes jet cavity interactions (see, for example, [2-4]), a jet impinging on a plate [1,5-9], open cavity flows [10-13], flows past snaked bodies [1, 14-16].

Numerical investigations of flows, containing the most number of “active” elements, allowed to find new self-oscillatory flow, namely flow near blunted bodies (cylinders or cones), giving off opposite jets [17,18]. This search for self-oscillatory flows is continued here. Jet flows near the bare open tube – inner cylinder are considered. These flows may contain secondary opposite ring jets. CFD studies of flows without significant secondary jets are carried out here. It is clear that if the cylinder radius \( r_{cyl} \) is equal to the inner radius of tube \( r_{tube} \), \( r_{cyl} = r_{tube} \), Hartmann whistle may be resulted. Similarly, if the cylinder radius \( r_{cyl} \) is much greater than the jet radius, the case of a jet impinging on a plate is resulted.

Since the purpose of recent investigations is to find new self-oscillatory regimes, jet radiiuses comparable with cylinder radiuses, are applied in CFD studies.

II. PHYSICAL AND COMPUTATIONAL MODELS

2.1 Turbulent viscosity calculations

The Reynolds-averaged Navier-Stokes equations added by an algebraic turbulent viscosity are used here and in previous author’s investigations [1,18-19]. A search for new self-oscillatory flows deals with trial calculations of numerous different flows. So, we need in a simple and universal procedure of turbulent viscosity calculations. Here the procedure [19], based on the Karman turbulent length scale formulae, is used.

This procedure deals with the Prandtl formulae \( \mu_{\text{tur}} = \rho |w| z^2 \), where \( w \) is vorticity, \( \rho \) is density,
\( l = k \), \( l \) is the turbulent length scale, \( k = 0.4 \) is the Karman coefficient. The Karman formulae \( l = k \frac{u}{\partial y} \), applied in the theory of a turbulent boundary layer, is generalized in [18-19]:

\[ l \approx |w(x,y)||[\nabla \omega(x,y)|, \quad (1) \]

where \( \nabla w(x,y) \) – vector \( \partial w / \partial x, \partial w / \partial y \), \( |\nabla w(x,y)| = [(\partial w / \partial x)^2 + (\partial w / \partial y)^2]^{1/2} \), \( w \) - vorticity.

Of course, the formulae (1) gives bad results in regions with “small” values of \( |\nabla w(x,y)| \) and so should be modified. Next procedure of turbulent viscosity calculations is used here:

\[ l = |w|/\sqrt{\nabla w^2} + \delta(|\nabla u|^2 + |\nabla v|^2)/|\nabla w| \] \quad (2)

\[ z = \min \{|(1+l^4/\lambda^4)(1+l^2(1.2 c^2)^2/\delta^6)|^{1/4}, d_w\} \] \quad (3)

\[ \mu_{tr} = \rho |w|(k)^2 \] \quad (4)

\( \lambda, \ c \) - control parameters, \( u, v \) – velocity components, \( d_w \) – distance to a solid surface, \( \delta = 0.0001 \) - the coefficient, providing regularity of written above formulas in the case \( \nabla w \approx 0 \), \( \text{Det} = \partial x/\partial \xi \times \partial y/\partial \eta - \partial x/\partial \eta \times \partial y/\partial \xi \), \( \xi, \eta \) – transformed variables. The geometrical parameter \( c \) should be equal approximately to a maximal width of shear layers or boundary layers, parameter \( \lambda < c \) allows to vary “level” of turbulence. To make the formulae (3) more clear the main expression of this formulae should be rewritten as a multiplication of two factors, \( l/(1+l^4/\lambda^4) \) \( \text{and} \) \( 1/(1+l^2(1.2 c^2)^2/\delta^6) \) \( \text{and} \) \( l/(1+l^4/\lambda^4)/(1+l^2(1.2 c^2)^2/\delta^6) \).

First factor approximates the Karman formulae in regions with small values of the relation \( ll/\lambda \). This factor increases asymptotically till the value \( l = \lambda \) as \( l \) increases. The second factor is approximately equal to 1 in the interval \( 0 < l < c \), then this factor quickly decreases and tends to the zero limit as \( l \) increases.

The turbulent length scale decreasing is intended to provide low level of the turbulent viscosity inside of circulation zones. Underlined expressions are averaged expressions in formulas (2)-(5), the averaging procedure consists of two steps

\[ f_1 = \int_{m\Delta}^{m\Delta} f(x+\xi,y) \Omega (m \Delta x,\xi) d\xi, \quad f_2 = \int_{m\Delta}^{m\Delta} f_1 (x,y+\xi) \Omega (m \Delta y, \xi) d\xi, \] \quad (5)

\[ \Omega (\epsilon, \zeta) = (1.2 - \zeta^2/\epsilon^2) / \int_{-\epsilon}^{\epsilon} (1.2-\zeta^2/\epsilon^2) d\zeta. \]

It should be noted, RANS have parabolic type, if turbulent viscosity does not depend on first or higher derivatives of solution functions. To eliminate dependence of turbulent viscosity (1)-(4) on first and second derivatives, the averaging procedure (5) is used. This procedure provides “smoothing” of turbulent viscosity, consequently, improves convergence to steady state solutions and prevents false unsteadiness. The most dangerous expression, containing second derivatives - \( |\nabla w|^2 \) - is averaged really twice, namely, averaging is used in formulas (2) and (4). It allows to decrease averaging region and to use the relatively small integer parameter \( m = 4 \) (see form. (5)), consequently, to get satisfactory resolution of the recent approach. Averaging is divided into two steps to diminish the computational cost of the procedure.

The purpose of usage of this model is to find flows, which keep unsteady regimes while the control parameter \( \lambda \) is increasing, but remains much less then geometrical lengths of problems. This
turbulence model may be considered as a tool to simulate approximately the turbulent diffusion influence on solutions of RANS.

2.2. Numerical method

An implicit conservative Runge-Kutta scheme [19] is employed here with some modifications. Initial method [19] is implemented in a computer code for sufficiently smooth curvilinear coordinate transformations \( x = x(a,b), \ y = y(a,b) \), mapping the unit square in the plane of variables \( a,b \) to a curvilinear quadrangle in the plane of physical variables \( x, y \). Within this approach, it is difficult to obtain satisfactory meshes for complicated physical domains. For that reason, a special version of the code is developed for the case when functions \( x = x(a,b), \ y = y(a,b) \) perform mapping of the unit square with excisions \( \{ 0 \leq a \leq a_0, 0 \leq b \leq b_0 \}, \{ a_1 \leq a \leq 1, 0 \leq b \leq b_1 \} \) to a curvilinear quadrangle with curvilinear quadrangular excisions (see figure 1). This version allows carrying out calculations, described below, without dividing complicated domains into subdomains. Both recent method and method [19] are third order in time and fourth order in space (viscous terms are approximated with second order).

![Figure 1. Schematic representation of a domain, a mesh and a jet.](image)

Naturally, numerical calculations deal with dimensionless variables. These variables are defined as relations of initial variables and next parameters of the outer stream or the body size: \( p_\infty \) - for pressure, \( \rho_\infty \) - for a density, \( \sqrt{p_\infty / \rho_\infty} \) - for a velocity, \( r_{tub} \) (the inner tube radius) – for space variables, \( r_{tub} / \sqrt{p_\infty / \rho_\infty} \) - for time.

2.3. Test calculations

Steady discontinued flow is studied numerically to prove that procedure of turbulent viscosity calculations does not produce false unsteadiness and turbulent low Mach number flow near plane surface is calculated to check the logarithmic low near plane surface. These test problems studies are written in [19] in detail.

The supersonic flow resulted from interaction of two parallel uniform streams (see figure 2) is calculated on the base of RANS equations added by the written above turbulent viscosity procedure. The flow is defined by upper stream parameters \( p^u = 1, \ \rho^u = 1, \ M^u = 2.4 \) and down stream parameters \( p^d = 0.25, \ \rho^d = 0.5, \ M^d = 4. \) The 180x135 grid is used. Figure 2a shows the density distribution in the case \( \mu_{turb} = 0 \). Figure 2b represents convergence histories of numerical flow fields. Karman viscosity parameters \( c = 0.075L \) (\( L \) - the left boundary size), \( \lambda = 0.05L \) are used. Calculations are carried out for CFL number 1.38, 1 - the ideal case \( \mu_{turb} = 0 \), 2 – the turbulent case, 3 – turbulent viscosity is calculated without averaging. So, convergence histories show, that averaging is important part of the recent turbulent viscosity calculations procedure since averaging provides convergence to a steady state solution. Figure
2b demonstrates quick convergence to a steady state solution of the method [20] for the CFL number 1.38. But since large CFL numbers may lead to increasing of computation errors in the case of unsteady flows, recent investigations, written below, are carried out for CFL numbers within limits 0.5 – 1.

![Figure 2. Streams interaction, a - the density distribution, b - convergence histories](image)

The low Mach number 2D flow near plane surface is calculated with usage of the Karman turbulent viscosity. Boundary conditions for computations are no-slip adiabatic wall on the plane solid surface, extrapolations on the outflow boundary, prescribed variables on the inflow edge. Namely, dimensionless inflow pressure and density are 1, the vertical velocity \( v \) is 0, the horizontal velocity is \( u = M_\infty \sqrt{\gamma} \sin \left( \frac{s\pi}{2} \right) \) if \( s = y/\theta \leq 1 \), \( u = M_\infty \sqrt{\gamma} \) if \( s = y/\theta > 1 \), \( \gamma = 1.4 \) – the specific heat ratio, \( \theta \) - the initial boundary layer thickness, \( \theta = 0.75h \). Calculations are carried out for flow conditions \( M_\infty = 0.3, \mu_\infty = 1.3e^{-6} \). Influence of turbulent viscosity on flow fields is dominant in this case. Karman viscosity parameters are chosen as \( c = 0.9\theta, \lambda = 0.6\theta \). The 451 \( \times \) 181 mesh is used.

![Figure 3. The flow near the plain surface, a – the computational domain, b - velocity profiles](image)

Figure 3a shows the computational domain and the mesh structure, figure 3b shows inflow and outflow velocity profiles. Since low value of inflow Mach numbers \( M_\infty = 0.3 \) and adiabatic wall condition result small compressibility of the flow, the logarithmic part of velocity profile near the wall should be formed. Really, the nearly rectilinear part of the outflow logarithmic profile is seen in fig. 3.

III. RESULTS AND DISCUSSION

3.1. Boundary conditions in jet impinging studies.

Figure 1 represents schematically a flow near a cylindrical body, placed in an open tube. There is a uniform sonic jet, outflowing from a nozzle at the left side of the picture. All variables are prescribed at inflow boundaries (\( HI,IA \)). The uniform stream with parameters Mach number \( M = 0.003 \), density \( \rho = 1 \), pressure \( p = 1 \) (in dimensionless form), is set at the \( IA \) boundary. The tangential velocity component is equal to zero and other variables are extrapolated at solid surfaces (\( CB,CD,FE,FG \)), which are represented in figure 1 by bold lines. The radial velocity component is equal to zero at the symmetry...
axis, other variables are extrapolated. Extrapolation conditions are used at the tube exit (DE). Extrapolation conditions are used at the AB boundary. Test calculations show, that numerical results are not sensitive to variation of this boundary place, if this boundary is sufficiently distant.

3.2. Unsteady flows near the pare cylinder – open tube.

Figure 4 shows density histories for two self-oscillatory flows, which are calculated for the nozzle exit temperature $\theta_{\text{jet}} = 0.8$ (in dimensionless form) and geometry parameters $r_{\text{cyl}} = 0.45$, $r_{\text{jet}} = 0.5$, $h_{\text{jet}} = 1.5$ (the distance of the nozzle exit section to the cylinder surface), $L_{\text{tub}} = 1.0$, $L_{\text{cyl}} = 1.0$ ($L_{\text{tub}}$ and $L_{\text{cyl}}$ - tube and cylinder lengths). The $515 \times 586$ mesh is used. Calculations are carried out for nozzle exit pressures $P_{\text{jet}} = 1.8$, $P_{\text{jet}} = 1.6$ and $P_{\text{jet}} = 1.4$. Karman viscosity parameters $c = 0.05R_{\text{tub}}$, $\lambda = 0.035R_{\text{tub}}$ are used in calculations of recent and next flows. Density magnitudes at cylinder edge, signed by arrow in figure 5, are plotted in figure 4 for the first two cases, a steady flow is received in the case $P_{\text{jet}} = 1.4$

Figure 4. Density histories for nozzle exit pressures $P_{\text{jet}} = 1.8$ and $P_{\text{jet}} = 1.6$.

Density histories, presented in figure 4, show that these flows are nearly periodic with $T = 2.68$ and $T = 2.07$ periods. Figure 5a represents the density distribution, calculated for $P_{\text{jet}} = 1.8$. Solid walls (cylinder and tube walls) are shown by bold lines. This figure corresponds to the final $t = 56.9$ instant (see figure 4).

Figure 5. Density distributions, a - $t = 56.9$, b - $t = 56.9 + T/2$.

To illustrate the flow dynamic through one period $T$ the density distribution for the instant $t = 57.1 + T/2$ is shown in figure 5b. Vertical arrows mark recent and previous (shown in figure 5a) shock wave positions. Significant oscillations of the shock wave position is seen in figures 5a and 5b.

Figure 6 shows density histories for the self-oscillatory flow, calculated for flow conditions $\theta_{\text{jet}} = 0.8$ $P_{\text{jet}} = 1.6$ and geometry parameters $r_{\text{cyl}} = 0.55$, $r_{\text{jet}} = 0.5$, $h_{\text{jet}} = 1.5$ (the distance of the nozzle exit section to the cylinder surface), $L_{\text{tub}} = 1.1$, $L_{\text{cyl}} = 1.0$ ($L_{\text{tub}}$ and $L_{\text{cyl}}$ - tube and cylinder lengths). The
515×586 mesh is used. Density magnitudes at points, signed by inclined arrows in figure 7a (see below), are plotted in figure 6. The upper graph corresponds to the first point, the down graph corresponds to the second point.

Density histories, presented in figure 6, show that this flow is nearly periodic with the $T=2.60$ period. Flow fields dynamics during one period after the final instant $t=61.7$ (see figure 6) is calculated. Density distributions are shown for instants $t=61.7$ (figure 7a) and $t=61.7+T/2$ (figure. 7b), correspondingly.

Vertical arrows mark positions of shock waves in figures 7a, 7b, recent and previous. It may be seen from these figures, that there is significant oscillations of the shock wave position, and the oscillations amplitude increases as the distance to axes symmetry increases. The faint secondary jet, going along the outward tube surface, is seen also in these figures.

**Figure 7. Density distributions, a - $t=61.7$, b - $t=61.7+T/2$.**

**IV. CONCLUSIONS**

Numerical search for new self-oscillatory compressible flows, started in [1,17-18], is continued in recent paper. Sonic underexpanded jet impinging on cylindrical bodies, placed in open tubes, is found to have self-oscillatory nearly periodic regimes. Flows without significant secondary opposite jets are considered here.

The flow near the pare open tube – inner body is defined by large number of control parameters and a search for self-oscillatory variants deals with large amount of the computational work. Simple approach RANS + the algebraic turbulent viscosity, based on the Karman length scale formulae, allows to make this search opportune since the procedure of turbulent viscosity calculations is fast and does not require complicated adjustment to new flow.
REFERENCES

[1] V. I. Pinchukov, “Numerical Modeling of Non-Stationary Flows with Transient Regimes”, Comput. Mathem. and Mathem. Physics, 49 (10), pp. 1844–1852, 2009.

[2] G Raman, E. Envia and T.J. Bencie, “Jet Cavity Interaction Tones”, American Institute of Aeronautics and Astronautics J., vol. 40 (8), pp. 1503–1511, 2002.

[3] S. Murugappan and E. Gutmark, “Parametric Study of the Hartmann–Sprenger Tube”, Experiments in Fluids, vol. 38 (6), pp. 813–823, 2005.

[4] J. Kastner and M. Samimy, “Development and Characterization of Hartmann Tube Fluid Actuators for High-speed Control”, American Institute of Aeronautics and Astronautics J., vol. 40 (10), pp. 1926–1934, 2002.

[5] W. Wu and U. Piomelli, “Large-Eddy Simulation of Impinging Jets with Embedded Azimuthal vortices”, J. of Turbulence, 16(10), pp. 44-66, 2014.

[6] C.-Y. Kuo and A. P. Dowling. “Oscillations of a Moderately Underexpanded Choked Jet Impinging Upon a Flat Plate”, J. Fluid Mech., vol. 315, pp. 267–291, 1996.

[7] Y. Sakakibara and J. Iwamoto, “Numerical Study of Oscillation Mechanism in Underexpanded Jet Impinging on Plate”, J. Fluids Eng., vol. 120, 477, 1998.

[8] G. E. Gorshkov and V. N. Uskov, “Specialties of Self- Oscillations, Arising from Interaction of Supersonic Underexpanded Jet with Finite Obstacle”, Prikl. Mekh. Tekh. Fiz., 40(4), pp. 143-149, 1999 (in Russian).

[9] B. Henderson, J. Bridges and M. Wernet, “An Experimental Study of the Oscillatory Flow Structure of Tone-Producing Supersonic Impinging Jets”, J. Fluid Mech., vol. 542, pp. 115-137, 2005.

[10] J. Rossiter, “Wind-Tunnel Experiments on the Flow over Rectangular Cavities at Subsonic and Transonic Speeds”, Technical Report Reports & Memoranda 3438, Aeronautical Research Council, 1964.

[11] C.-J. Tam, P. D. Orkwis and P. J. Disimile, “Comparison of Baldwin-Lomax Turbulence Models for Two-Dimensional Open-Cavity Calculations”, AIAA J., vol. 34(3), Technical Notes, pp. 629–632, 1996.

[12] C.-J. Tam, P. D. Orkwis and P. J. Disimile, “Algebraic Turbulence Model Simulations of Supersonic Open-Cavity Flow Physics”, AIAA J., vol. 34(11), pp. 2255-2260, 1996.

[13] N. Murray, E. Sillström and L. Ukeiley, “Properties of subsonic open cavity flow fields”, Physics of Fluids, vol. 21, 095103-16, 2009.

[14] M. Gauer and A. Paull, “Numerical investigation of a Spiked Nose Cone at Supersonic Speeds”, Journal of Spacecraft and Rockets, vol.45, N. 3, pp. 459-471, 2008.

[15] W. Caarese and W. L. Hankey, “Modes of Shock Wave Oscillations on Spike Tipped Bodies”, AIAA Journal, vol. 23, N. 2, pp. 185-192, 1985.

[16] R. C. Mehta, “Pressure Oscillations Over a Spiked Blunt Body at Hypersonic Mach Number”, Computational Fluid Dynamics Journal, vol.9, N. 2, pp. 88-95, July 2000.

[17] V. I. Pinchukov, “Modeling of Self-Oscillations and a Search for New Self-Oscillatory Flows”, Mathematical Models and Computer Simulations, vol. 4(2), pp. 170–178, 2012.

[18] V. I. Pinchukov, “Self-oscillatory Flows near Blunted Bodies, Giving off Opposite Jets: CFD Study”, Intern. J. of Engineering and Innovative Technology, vol. 6, Issue 5, pp. 41-46, 2016.

[19] V. I. Pinchukov, “Karman Turbulence Model Simulations of Supersonic Open Cavity Flows”, Intern. J. of Innovation in Science and Mathematics, vol. 5, Issue 3, pp. 89-94, 2017.

[20] V. I. Pinchukov, “Numerical Solution of the Equations of Viscous Gas by an Implicit Third Order Runge-Kutta Scheme”, Comput. Mathem. and Mathem. Physics, vol. 42(6), pp. 898-907, 2002.