Physical Science International Journal
11(1): 1-11, 2016, Article no.PSIJ.26624
ISSN: 2348-0130

SCIENCEDOMAIN *international*
www.sciencedomain.org

A New Approach in Interpretation and Solving Certain Hydrodynamic Phenomena and Systems

Andrei Nechayev[1*]

[1]*Department of Geography, Lomonosov Moscow State University, 119991, Leninskiye Gori 1, Moscow, Russian Federation.*

Author's contribution

The sole author designed, analyzed and interpreted and prepared the manuscript.

Article Information

DOI: 10.9734/PSIJ/2016/26624
Editor(s):
(1) Yang-Hui He, Professor of Mathematics, City University London, UK And Chang-Jiang Chair Professor in Physics and Qian-Ren Scholar, Nan Kai University, China; & Tutor and Quondam-Socius in Mathematics, Merton College, Univ. of Oxford, UK.
(2) Stefano Moretti, School of Physics & Astronomy, University of Southampton, UK.
Reviewers:
(1) Anonymous, University of St. Thomas, USA.
(2) Kuei-Hao Chang, National Cheng-Kung University, Taiwan.
(3) Kunjan Shah, C. U. Shah University, Gujarat, India.
Complete Peer review History: http://sciencedomain.org/review-history/14987

Original Research Article

Received 26th April 2016
Accepted 4th June 2016
Published 11th June 2016

ABSTRACT

A new theoretical description of a number of hydrodynamic phenomena and paradoxes (such as laminar-turbulent transition, Magnus effect, Einstein's tea leaves paradox, drag mechanism, wing lift, Karman vortex Street) is proposed. This description is based on the fundamental equations of hydrodynamics and corresponding detection of low and high pressure zones in the given fluid flow.

Keywords: Hydrodynamic paradox; laminar-turbulent transition; drag mechanism; Magnus effect; Karman Vortex Street.

1. INTRODUCTION

A characteristic feature of classical hydrodynamics is the presence of a strong, well-developed mathematical apparatus that allows to confirm many experimental results (mainly by numerical simulation), and the simultaneous absence of a clear understanding at the level of "simple physics" of a number of hydrodynamic phenomena. The consequence of this fact is a lot of the so-called hydrodynamic paradox [1], demonstrating to the satisfaction of pupils and students "miracles" of the flowing fluid but do not receiving a satisfactory physical interpretation. Suffice it to mention the two "iconic" phenomena: the laminar-turbulent transition and the lift of the

**Corresponding author: E-mail: and.nechayev@gmail.com;*

wing. In the first case, the critical conditions are well known, empirically established, 3D-modeling of the transition to turbulent flow regime is successfully implemented, but the deduction of the critical Reynolds number even in its simplest form in the classical theory is missing. In the second case, to explain the effect and to facilitate the calculations the existence of circulation and of the so-called bound vortex around the wing was postulated, although their reality is not confirmed experimentally. It can be assumed that the fundamental laws of hydrodynamics are true, but their use in specific situations is not always correct. This is especially regard events, where an important role is played by the boundary conditions.

1.1 Basic Equations and Concept

Here are the basic equations of hydrodynamics [2]. First of all, the Navier-Stokes equations, which represents the second law of Newton, recorded for an elementary volume of fluid: the rate of change of momentum is equal to the sum of all forces acting on the mass:

$$\frac{D(\rho\vec{v})}{Dt} = \vec{F} - \nabla p + \mu\nabla^2\vec{v} \tag{1}$$

where $\vec{v}$ is the speed, p is pressure, ρ is density, μ is viscosity of the fluid at a given point, $\vec{F}$ is a mass force, ∇ is gradient, ∇^2 is Laplacian. In the absence of mass forces, with zero viscosity and constant density, equation (1) is converted to Euler's equation, which in one-dimensional case (for the current line with a coordinate s) is as follows:

$$\rho(\partial v/\partial t + v\,\partial v/\partial s) = -\partial p/\partial s \tag{2}$$

The integral on the current line in the stationary case ($\partial v/\partial t = 0$) gives the Bernoulli equation:

$$p + \rho\frac{v^2}{2} = const \tag{3}$$

An integral part of fluid dynamics (and the Navier-Stokes equations) is the continuity equation:

$$\partial\rho/\partial t + div\rho\vec{v} = 0 \tag{4}$$

In the current tube with section S the continuity equation gives the condition of constant mass flow:

$$\rho S v = const \tag{5}$$

The system of equations must include the equation of state for compressible fluids (gases):

$$p = \rho RT \ , \tag{6}$$

where R is the universal gas constant, T is a temperature.

Completeness of a theoretical description is provided by the boundary and initial conditions, the importance of which can't be overestimated. For example, on the surface of the fixed solids the fluid velocity must always be equal to zero.

In accordance with the classical concepts [2,3], we will use the terms "critical point" and "boundary layer", which are introduced for areas of flow-collision with fixed boundaries or obstacles. Thus, in case of a fluid falling at right angle to a flat surface (Fig. 1), the pressure near the critical point K (where $v \to 0$) in accordance with the equation (3) tends to a maximum value equal to $\rho v^2/2$.

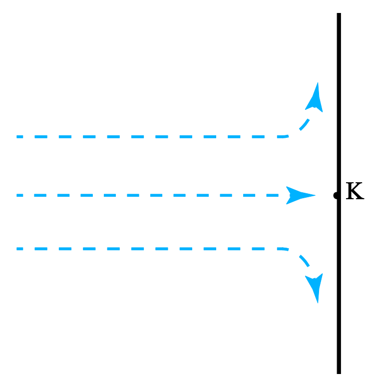

Fig. 1. Fluid flow impinging on a flat surface at right angle, spreads to the left and right of the critical point K, where pressure is maximum

Principal aspect of the concept of "critical point" does not change depending on what is the nature of the flow: Laminar or turbulent. In the latter case, obviously, the initial average speed of individual jets subjected to the collision with the surface is close to v, and it can be expected that the pressure near the critical point will fluctuate correspondingly near the value $\rho v^2/2$. To ensure evacuation of fluid from the region of jet-plane collision, the fluid pressure along the solid surface should decrease to the left and to the right from the point K.

In case of fluid motion parallel to the solid surface a boundary layer with thickness δ forms, and some perturbed streams reach the area where fluid may experience a deceleration to zero speed. This occurs, for example, at the critical point K (Fig. 2), the pressure near it increases in accordance with (3), and the jet is "reflected" from the wall, choosing a new direction depending on the magnitude of a local pressure increase and the total pressure distribution in the flow in the vicinity of this point. The nature of possible interaction of the jet, colliding a solid surface, Fig. 3 illustrates, where an enlarged scale of supposed surface structure is shown. Streams or jets decelerate at the micro-irregularities having a corresponding plane oriented upstream. On these flat planes the zones of local high pressure, proportional $\rho v^2/2$, are formed.

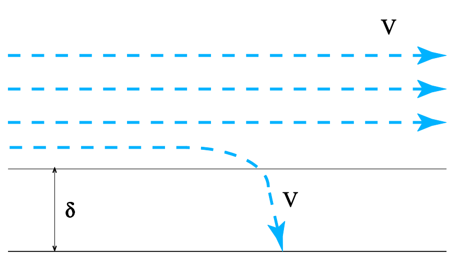

Fig. 2. Deviation of the individual jet in the direction of the boundary layer and solid surface. At critical point K the jet decelerates

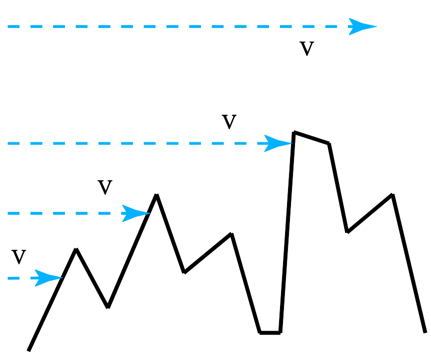

Fig. 3. Supposed collision of fluid jets with a solid surface on the crystal structure level (highly magnified)

2. THEORETICAL RESULTS AND DISCUSSION

Let's see some "iconic" hydrodynamic phenomena and paradox, which can be interpreted and calculated with ideas on the decelerating of individual streams and jets near solid surfaces.

2.1 Einstein's Paradox: The Tea Leaves Accumulate in the Bottom Center of the Glass

The phenomenon, which has attracted the attention of the great physicist, was on everyone's mind when there were not tea bags. If a glass of tea contains tea leaves, then after stirring sugar with a spoon all tea leaves accumulate at the bottom of a glass at its center. This experiment is easy to repeat in either cylindrical vessel (e.g., in a pan), filling it with water and pouring into water tea leaves or any powder having a density slightly higher than the density of water. When water is put in rotation, the tea leaves realize in its volume a corresponding circular motion. During the deceleration of water by the walls, tea leaves sink to the bottom and actively, as if some mysterious force wings them, rush to the center, where they are accumulated in an expressive hillock!

Einstein himself gave an explanation of his paradox [4]. He suggested that the rotating fluid particle is subjected to the centrifugal force, which initially is balanced by the pressure gradient directed towards the center of rotation. By the way, due to this gradient, the free surface of a rotating liquid has a parabolic shape. Near the bottom where the fluid is decelerated, its linear velocity decreases, and decreases the centrifugal force. A pressure gradient is the same as for the higher layers of the fluid. Therefore, in the bottom layers the centripetal flow occurs, -- Einstein concluded. However, a simple experiment is able to refute Einstein's version. It is enough to replace the water of a much more viscous fluid, such as sunflower oil and the paradox disappears: tea leaves accumulate nowhere. The viscosity of the oil is almost 40 times higher than the viscosity of water, and the oil rotation in the glass is laminar rather than turbulent. How this fact can explain Einstein's paradox?

In laminar rotation all current lines are plane-parallel, the turbulent flow consists of the jets which are deflected upward and downward of rotation direction and may face irregularities of the bottom surface as shown in Figs. 2, 3. Assuming that the water in the vessel is rotated with an angular velocity Ω, each particle including particles near the bottom of the vessel have a linear velocity $V = \Omega r$, where r is the radius of the particle location. Consequently, the jet, deviating from its circular path overcomes a boundary layer, reach the surface of the bottom and create at the place of collision a local increase in pressure equal $\rho\Omega^2 r^2/2$, which is greater, the larger the radius of rotation r. The corresponding pressure gradient, providing the appropriate force, is directed to the center and is equal $\rho\Omega^2 r$. It decreases at the deceleration of fluid rotation, remaining non zero, and is present only in the vicinity of the bottom surface. Therefore the centripetal movement of tea leaves develops, when they begin to touch the bottom (this is clearly seen in the experiment in a wide pan). While Ω is large and tea leaves are suspended, there are no aspiration to the center.

2.2 Magnus Effect

Let's analyze the well-known hydrodynamic phenomenon that occurs when fluid flows around a rotating body. If the body has a circular shape and is not fixed (for example, a football or a tennis ball), it is deflected due to its rotation to the direction perpendicular to the flow. There is a Magnus force, which is explained by the difference in velocity of fluid at the top and bottom of the body (Fig. 4) and by the Bernoulli equation: pressure is less where the flow and rotation velocities are summed. The correctness of the use of Bernoulli's law in this version is not obvious. Let us show how to interpret the Magnus effect, considering only the deceleration of jets on the surface of a rotating body.

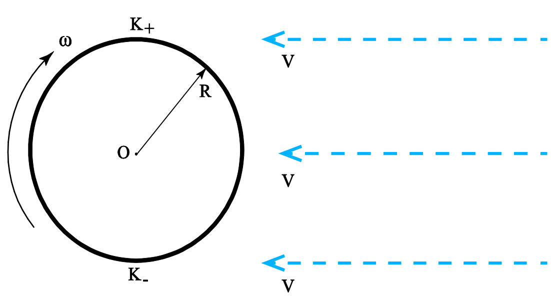

Fig. 4. To illustrate Magnus effect: A rotating body and the incoming fluid flow

At the point K_+ the incoming flow meets the surface of a rotating body with overall speed $(V + \omega R)$. There is, probably, as shown in Fig. 3. At critical points (in the coordinate system associated with the body) fluid velocity falls to zero and the pressure in accordance with Bernoulli's equation (3) is increased by an amount equal $\rho(V + \omega R)^2/2$. At the bottom point K_- the velocity of collision of body surface and the flow is equal to $|V - \omega R|$ and a corresponding increase in pressure is $\rho(V - \omega R)^2/2$. Obviously, it is smaller by the value $2\rho V \omega R$ than at the top point K_+. The maximum pressure difference will exist between the upper and lower points. At the remaining points of the upper half of the body facing the flow, the overpressure as a result of fluid decelerating will also be larger than at the bottom half. Thus, the Magnus force, in our view, is really due to Bernoulli's law in the part where it connects local fluid decelerating and the corresponding pressure increase.

2.3 Laminar-turbulent Transition

The effect of transformation of flow velocity in a local increase in pressure in the zones of deceleration of the individual jets (at the boundaries and obstacles) is able to originate reverse currents and eddies. The result can be a laminar-turbulent transition. For the first time this mechanism for Poiseuille flow in a pipe has been proposed in [5]. We reproduce below the basic idea of this analysis, and then apply it to the case of laminar-turbulent boundary layer transition.

We assume that in any laminar flow, there are disturbed streams caused by a variety of irregularities or obstacles, for example, for the pipe flow the conditions of the entrance of fluid into the inlet are principal. Here, as in [5], we consider the heterogeneity as an annular aperture with width h, similar to that used in experimental study [6] (Fig. 5).

We assume that the fluid streams at a velocity $u(h)$ deviate the ring diaphragm and face the opposite wall of the pipe. For simplicity and clarity, we assume that isolate fluid jet moves as a single body, and on its way to the collision with the wall is not experiencing any of viscous friction or resistance. So at the point K of collision with the wall where $u = 0$ the fluid pressure in the jet will rise by the value $\rho u^2(h)/2$. The reducing of the flow pressure at distance from the diaphragm to the point K is equal $\frac{dp}{dx}\frac{2R-h}{tg\alpha}$. If the overpressure at point K exceeds this pressure drop, it is possible to reverse the flow from the point K back to the diaphragm, which can be transformed into the vortex. If we denote the relative magnitude of heterogeneity k, where $k = h/R$, the condition of the appearance of counter-flow can be obtained for the Poiseuille flow and the corresponding critical Reynolds number can be derived [5]:

$$\mathrm{Re} \equiv \frac{\rho 2RU}{\mu} > \frac{8}{k^2(2-k)tg\alpha} \equiv \mathrm{Re}_{cr} \qquad (7)$$

where U is the average velocity of the flow, μ is viscosity of the fluid, α is an average angle of jet deflection. Note that for small inhomogeneity $(h << R, k << 1)$, we obtain the dependence of $\mathrm{Re}_{cr} \sim h^{-2}$, which was discovered experimentally [6].

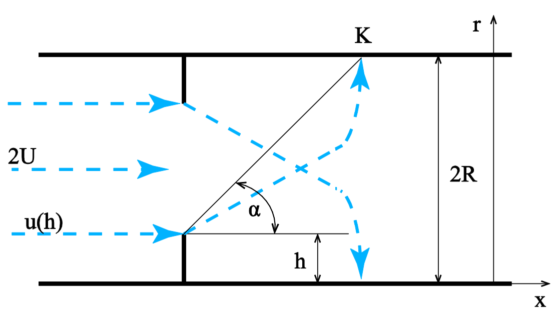

Fig. 5. Poiseuille flow in a pipe with radius R. The jets are deflected by ring diaphragm with width h

In the case of boundary layer a similar approach may be applied. Let there be a steady flow of fluid along the plane solid surface and the boundary layer with thickness δ (Fig. 6).

To calculate the critical Reynolds number, it is necessary to estimate the pressure gradient $\partial p/\partial x$, which provides steady flow of fluid in the x direction. We use the classical approach. We mark out in the direction of flow in the boundary layer one fluid strip with length l, width d and thickness δ. It is subjected to the force equal $\frac{\partial p}{\partial x} d\delta l$ which counterbalances the force of viscous friction, equal $\mu\frac{U}{\delta}ld$. Equating both forces, we obtain the pressure gradient: $\frac{\partial p}{\partial x} = -\mu\frac{U}{\delta^2}$ that naturally agrees with Navier-Stokes equation (1). The pressure increase at critical point B due to stream decelerating is $\rho U^2/2$, and the corresponding decrease of the pressure in the flow at distance AB is equal to $\frac{\partial p}{\partial x}\frac{\delta}{tg\alpha}$. Consequently, we obtain the condition of reverse-flow with the critical Reynolds number determined by the thickness δ of boundary layer:

$$\mathrm{Re} \equiv \frac{\rho\delta U}{\mu} > \frac{2}{tg\alpha} \equiv \mathrm{Re}_{cr} \qquad (8)$$

Thus, the threshold of the onset of turbulence, that is the critical Reynolds number, in the case of the boundary layer is determined by the level of flow disturbance, characterized by the deflection angle α.

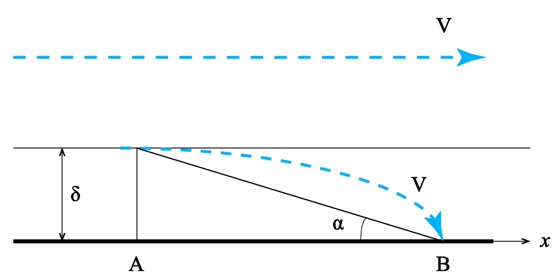

Fig. 6. The flow along a flat surface with an individual stream deflected in the boundary layer with thickness δ

2.4 The Flow in the Pipe

Any real fluid flow - whether the flow of water or air - always starts somewhere and ends somewhere. Accelerated zones throughout the region are replaced by the deceleration zones. The pressure in the flow responds to the flow rate factor in accordance with a whole set of laws and equations. Among them the Bernoulli law perhaps the main but not the only one. Two laws "work in tandem" in the hydrodynamic phenomena: Bernoulli's law and the law of mass conservation, which are embodied in the corresponding equations. Proper and correct application of these laws allows to solve many of the hydrodynamic paradox. To better understand the physical nature of the hydrodynamic equations, consider a trivial, at first glance, the flow of fluid in the tube (Fig. 7).

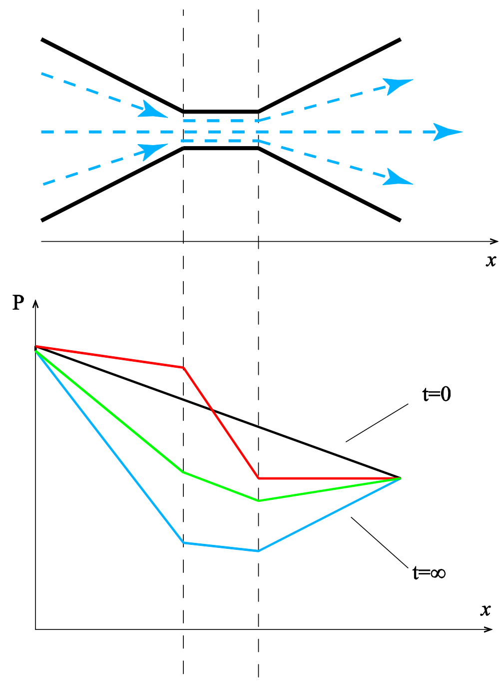

Fig. 7. The fluid flow in a pipe with a contraction and expansion. The pressure on the inlet and outlet of the pipe is kept constant. The lower graph is the expected pressure distribution at different times. Black color is the pressure at the initial moment. Blue color is the pressure in a steady stream

For the initial time ($t=0$) the pressure drop is constant along the entire pipe (it is set with the speed of sound). With the acceleration of fluid its pressure begins to change. Initially, all fluid moves at the same speed v, but the total flow $\rho v S$ (where S is the section) throughout has to be different, it is equal to the mass of fluid flowing in one second through a given section. Hence, in the narrowing portion of the tube, where S decreases, the fluid tends to compress and it tends to expand in the expanding part. Therefore, in the left side of the pipe the pressure increases, and in the right it falls (red line on Fig. 7). The pressure drop in the narrow part of the tube increases, so fluid is accelerated. The pressure in the expanding part is aligned, the fluid slows down there. Next, the Navier-Stokes equations and the continuity equation are adjusted to each other, the pressure reduction factor in the narrow part of the tube plays a leading role. As a result the pressure gradient in the left part increases to accelerate fluid to "slip" through the narrow section of pipe, and pressure gradient on the right side changes sign to decelerate fluid and make its speed in accordance with the flow in the wide part of the tube (green and blue line, Fig. 7). Strictly speaking, the Bernoulli equation in this case is not correct to use, because of the presence of frictional forces, but it gives a good quality estimation if the flow through the tube is not changed. The speed in this case is inversely proportional to the cross section of the pipe, and a stationary pressure will correspond to $\rho v^2/2$. If, for example, a narrow section S where the flow velocity is V, is followed by the extension of pipe cross section until S_0, the expected pressure drop Δp in the narrowest part will be equal (ignoring friction):

$$\Delta p = \frac{\rho V^2}{2}[(\frac{S}{S_0})^2 - 1] \quad (9)$$

Equation (9) is valid for an incompressible fluid, when the frictional losses (laminar or turbulent) are negligible. The air friction is small, but the air density is directly related to its pressure via the equation of state (6), which, strictly speaking, can not be ignored. Let's illustrate these particularities in the theoretical analysis of the well-known hydrodynamic paradox: "the behavior of the paper cone in a funnel with an air blowing [1]."

If you put inside of the glass funnel a paper cone of the appropriate form (Fig. 8) and blow into the tube, the air stream does not eject a cone out: On the contrary, the cone is drawn into the funnel even if the last is directed down.

Let the air flow entering the funnel tube to be constant and equal M. The same flow is stored inside funnel, where its cross section has an annular shape and is approximately equal to $\pi(r+\Delta R)^2 - \pi r^2 = 2\pi r \Delta R + \pi(\Delta R)^2$ (Fig. 8). The pressure at the outlet of the funnel is equal to $p_0 = \rho_0 RT$, the average air velocity is V_0. At the apex of the cone $(s=0)$ the corresponding flow parameters are equal p_1, ρ_1, V_1. Consider the movement of air in the absence of friction along the current line s (Fig. 8). Integrating the equation (1) and using equations (4) - (6), we obtain the Bernoulli equation in our case:

$$RT\int \frac{d\rho}{ds}ds + \int \frac{d(\rho v)}{ds}vds = 0 \qquad (10)$$

Given that $r = s\sin\alpha$, we can determine the air density and velocity using a system of two equations (11)-(12) that can be solved iteratively.

$$\rho = \rho_0 + 1/RT\int \frac{\partial(\rho v)}{\partial s}vds \qquad (11)$$

$$\rho v = \frac{M}{2\pi s\sin\alpha\Delta R + \pi(\Delta R)^2} \qquad (12)$$

From these equations we can deduce, that the pressure on the side surface of the paper cone is always less than atmospheric pressure p_0 as $\frac{\partial(\rho v)}{\partial s} < 0$, and its maximum drop (near the apex) will be the greater, the larger square of the velocity V_1, the smaller ΔR and the larger outlet radius R_0.

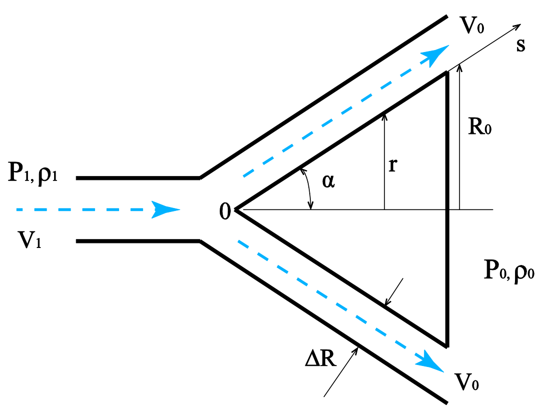

Fig. 8. The funnel with a paper cone inside. Air enters the funnel tube and flows through the gap with width ΔR between the walls of the funnel and a paper cone

2.5 The Drag Mechanism

When fluid flows around the obstacle, it surrounds it with pressure, and much depends on how this pressure is distributed. The plane rises into the air, dolphin catches ship, soccer ball flies into the goal, a tornado tears off the roof. The behavior of the fluid is always natural. Let's try to understand how these hydrodynamic laws can work in the simplest cases.

Let's start with the flow around a flat plate (Fig. 9).

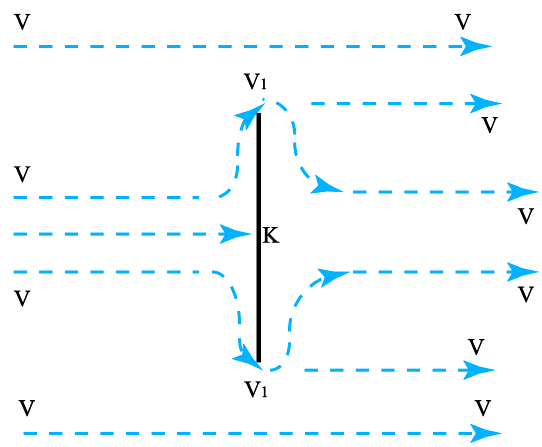

Fig. 9. A plate streamlined with a constant flow of average speed V. At the edges of the plate speed is increased. At the critical point K the pressure is maximum

The incoming fluid flow goes around the plate. In order to satisfy the mass conservation law, it should accelerate at the edges of the plate as if the flow is narrowed and $V_1 > V$. Behind the plate the flow begins to spread, and at some distance from the plate the velocity of all jets becomes equal to V again. Consequently, the speed of jets passing near the center of the plate at first increases and then decreases. This process can be provided by reducing the pressure on the edges of the plate (due to the reasons discussed above in the case of fluid flow in the narrow and expanding pipe section). Excessive fluid pressure in the center of the plate, at a critical point *K*, where the incoming stream is decelerated to zero velocity, contributes to the fluid acceleration, but a major role is apparently plays the expansion factor behind the plate: The flow here is obliged to decelerate as $V_1 > V$, and it is caused by the negative pressure gradient. According to Bernoulli's equation (it is true for jets, flowing stationary around the plate) pressure increase in the flow direction behind the plate should be equal $(\rho V_1^2/2 - \rho V^2/2)$. Since the pressure does not change at the cross section, because there are no transversal motion here, the pressure drop at the total back side of the plate remain the same with respect to the cross section where the flow restores its speed V (at the end of the "trace"). The magnitude of pressure drop is greater, the greater the velocity V_1, which increases with the transverse size of the plate. The total drag should obviously include the increasing pressure before the plate (it is proportional to $(\rho V^2/2)S$, where S is the area of the plate) and the pressure drop behind the plate due to the spreading of the stream.

Let's see what happens to the pressure and drag distribution when changing the shape of the obstacle body and take a hemispherical object, streamlined by fluid flow (Fig. 10).

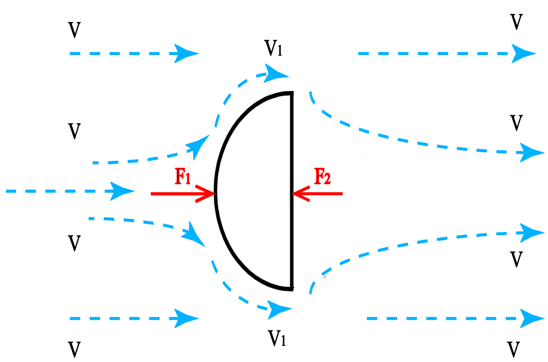

Fig. 10. Fluid flow around hemisphere. Red arrows indicate the pressure forces. Behind the body pressure considerably less because of the spreading flow

The stream flowing around a hemisphere is accelerated by the pressure drop, which decreases from the center (where it has the maximum value $\rho V^2/2$) to the edges. This pressure acts perpendicular to the body surface, but for the total drag the components along the direction of flow are only essential, and they reduce to zero towards the edges of the hemisphere. Therefore, the pressure force (Fig. 10) in the case of the hemisphere will always be less than the equivalent force in the case of a plate of the same cross section. The pressure reducing behind the hemisphere defines the force F_2 and depends on the magnitude of V_1 which can be smaller than in the case of the plate. As a result, the total drag $(F_1 - F_2)$ is less for the hemisphere than for the plate. It will be even less if the rear part of the body obtain a conical shape (Fig. 11).

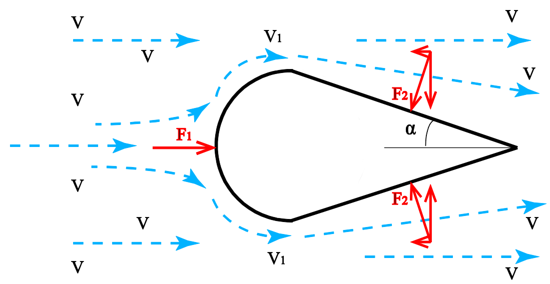

Fig. 11. Flowing around the conical shape body. The pressure forces F_2 on the rear of the body have components against the stream and totally reduce the drag

The maximum reduction in pressure in the flowing stream occurs at the beginning of the flow spreading behind the body and is equal $\Delta p = (\rho V_1^2/2 - \rho V^2/2)$. It is reduced to zero by the end of the body. If we take the average reduction of pressure on the back side surface of the body equal $\Delta p/2$, the overall reduction in the pressure force (per unit length of the surface in perpendicular direction to the flow) will be equal to $\frac{\Delta p}{2}\frac{R}{\sin\alpha}$ (where R is the radius of the hemisphere), and the reduction of its upstream component will be respectively $\frac{\Delta p}{2}R$ (Fig. 11). Decrease of the pressure force behind the body means a corresponding increase in the force of drag. For the body with a hemispherical shape (Fig. 10) reducing of the pressure Δp occupy the entire rear surface of the body and a corresponding decrease force gives ΔpR, which is twice more than for the conical "tail" body where the reduction in pressure is distributed across the all back side surface.

Thus, in all cases, reduction in pressure behind the body and a corresponding increase in drag will be greater, the greater the flow velocities V_1 and V. The connection between them can be written in the form of $V_1 = aV$, where the coefficient $a > 1$ is determined by the transverse dimensions and shape of the streamlined body.

2.6 The Lifting Force of the Wing

The approach to evaluation of pressure forces acting on the streamlined body proposed above allows us to explain the origin of the wing lifting force. Consider the conventional image of the wing with a distinctive asymmetrical profile (Fig. 12). The angle of attack is zero. The stream runs onto the front part of the wing, air flows around the profile, narrowing and accelerating until V_1, and further expands, reducing its speed to V. From the "drag" of the above section, it follows that above the wing there is a low pressure zone with respect to the atmospheric pressure of the ambient air. The average value of pressure reduction, proportional to $\rho(V_1^2 - V^2)/4$, multiplied by the area of the wing and $\cos\beta$, creates the desired lifting force and, multiplied by $\sin\beta$, gives addition to the drag force.

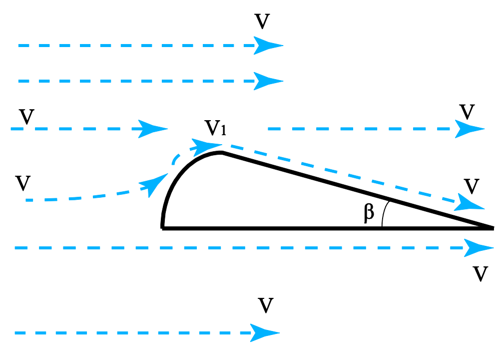

Fig. 12. The origin of wing lift at zero angle of attack

If the wing has the angle of attack α, the flow has somewhat different form (Fig. 13), but the fundamental cause of the lift remains the same, it is increased by the additional flow pressure on the lower part of the wing, which can be estimated by the decelerating to zero of the velocity component perpendicular to the bottom surface of the wing.

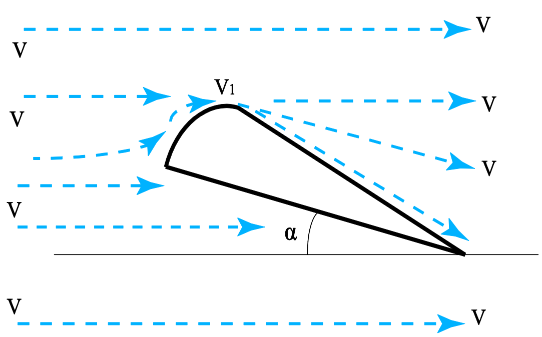

Fig. 13. Streamlining of the wing with the angle of attack α

This reduction gives the corresponding increase in pressure, proportional $\rho(V\sin\alpha)^2/2$, and the corresponding contribution to the lifting force proportional $\rho\cos\alpha(V\sin\alpha)^2/2$. The main contribution to the lifting force provide the decrease in air pressure in flow spreading over the wing. The magnitude of this reduction gives the integration of $\rho(V_1^2 - V^2)/2$, which is determined by the value of V and streamline coefficient a coming from the relation $V_1 = aV$. The last also depends on the angles α and β, as they determine the flow velocity V_1.

2.7 Ping-pong Ball in the Air Stream

The approach used above makes it easy to explain the well-known paradoxical behavior of a ping-pong ball in the air stream [1]. The ball "sticks" to the air stream and keeps it not only in a vertical position, but also when the air jet is inclined to the horizon (Fig. 14). The jet flows around the ball on one side, in streamlined place the flow velocity increases up to V_1, behind the ball the spreading zone is formed and the pressure decreases with respect to the atmospheric one. The maximum of this reduction is equal to $\rho(V_1^2 - V^2)/2$. On the opposite side the atmospheric pressure pushes to the ball and makes it "stick" to the jet.

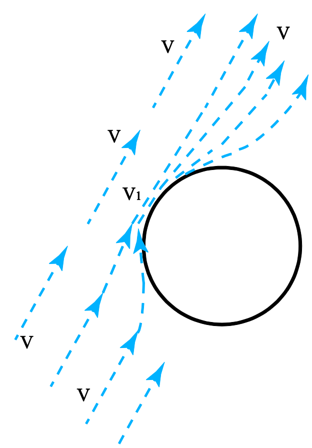

Fig. 14. The ball streamlined by the air jet directed at an angle to the horizon

2.8 Von Karman Vortex Street

Finally, we demonstrate how our approach makes it possible to interpret the hydrodynamic phenomenon, known as "Karman Vortex Street".

Consider steady stream of fluid with average velocity V flowing around the obstacle with cylindrical shape, which axis is located perpendicular to the flow (Fig. 15). Fluid flows around the cylinder, being accelerated at its edges. Assume that the accelerated flow area (with an average speed $V_1 > V$) form a kind of "stripes" with thickness d. Restoration of flow velocity V behind the cylinder takes place at a distance L equal to the length of the "trace".

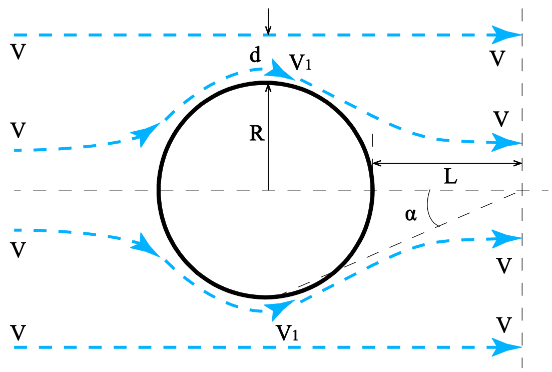

Fig. 15. A cylindrical obstacle with the axis perpendicular to the flow is streamlined by the fluid

Thus, the fluid stream with the thickness $2R$, flowing around a cylinder, is divided into two bands each with thickness d. In accordance with the continuity equation $RV \approx dV_1$. The pressure gradient in the flow behind the cylinder can be estimated using the classical procedure. The pressure force on the strip of unit length (along the axis of the cylinder), of thickness d and length $(L+R)$ is equal to $\frac{dp}{dx}(L+R)d$ (where the axis x is directed along the flow). It is balanced by the force of viscous friction, since the flow velocity is reduced to zero at the center line, dividing the cylinder (Fig. 15). This force can be overestimated by the value $\mu \frac{V_1}{R}(L+R)$. The equality of these two forces and conditions of continuity give us: $\frac{dp}{dx} = \frac{\mu V_1^2}{R^2 V}$. We assume for simplicity again that $V_1 = aV$ where $a > 1$. So then $\frac{dp}{dx} = \frac{\mu a^2 V}{R^2}$. We now estimate the magnitude of pressure reduction on cylinder edges Δp_1^- caused by the spreading of the flow and reducing its speed from V_1 until V. It is easy to see that in accordance with the Bernoulli law it is: $\Delta p_1^- = \frac{\rho}{2} V^2 (a^2 - 1)$ (viscous friction between adjacent streams we have neglected). Standard pressure drop behind the cylinder Δp_2^-, providing fluid flow, obviously, is: $\Delta p_2^- = \frac{dp}{dx}\frac{R}{tg\alpha} = \mu \frac{a^2 V}{R tg\alpha}$. If $|\Delta p_1^-| > |\Delta p_2^-|$, the flow in opposite direction (counter-flow) towards each of the cylinder edges becomes possible, as observed in the experiment with Karman vortex street. This condition can be written as $\frac{\rho}{2} V^2 (a^2 - 1) > \mu \frac{a^2 V}{R tg\alpha}$, or $\mathrm{Re} \equiv \frac{\rho V R}{\mu} > \frac{2a^2}{(a^2 - 1) tg\alpha}$. Because of the asymmetry between the streams around the cylinder edges which is always present in real situation, one of the sides gets the advantage for the beginning of the reverse flow and further formation of a vortex. From this the alternation of the vortices takes its origin.

3. BRIEF CONCLUSIONS

For the correct interpretation of hydrodynamic phenomena at the interaction of the fluid stream with a streamlined solid body, you can use the Bernoulli and continuity equations to locate the zones of high and low pressure. The first are related to the flow deceleration near critical points and to the places of the flow narrowing. The second are caused by the flow deceleration in the places of spreading. The interaction of these high and low pressure zones transforms the fluid flow, causing the diversity of hydrodynamic phenomena.

COMPETING INTERESTS

Author has declared that no competing interests exist.

REFERENCES

1. Girke R, Sprockhoff G. Physikalische schulversuche. Flussigkeiten und Gase Berlin; 1956.

2. Prandtl L. Essentials of fluid dynamics. Blackie & Son, London; 1952.
3. Schlichting H, Gersten K. Boundary layer theory. Springer, Berlin, Heidelberg, New York; 2000.
4. Einstein A. The cause of the formation of meanders in the courses of rivers and of the so-called baer's law. Die Naturwissenschaften. 1926;14.
5. Nechayev A. Theoretical conditions of pipe flow transition: New approach to the old problem; 2014. Available:htpp//arXiv:1409.1404[physics.flu-dyn]
6. Nishi M, Unsal B, Durst F, Biswas G. Laminar-to-turbulent transition of pipe flows through puffs and slugs. J. Fluid Mech. 2008;614:425-446.

Peer-review history:
The peer review history for this paper can be accessed here:
http://sciencedomain.org/review-history/14987