REPORT DOCUMENTATION PAGE

Form Approved OMB No. 0704-0188

1. AGENCY USE ONLY (Leave blank)

2. REPORT DATE

September 2000

3. REPORT TYPE AND DATES COVERED

Conference Proceedings

4. TITLE AND SUBTITLE

Organized Vortical Motion As A Basis For Boundary-Layer Control International Workshop

5. FUNDING NUMBERS

F61775-00-WF041

6. AUTHOR(S)

Conference Committee

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

Institue of Hydromechanics, National Academy of Sciences
8/4 Zheliabov St.
Kiev 04057
Ukraine

8. PERFORMING ORGANIZATION REPORT NUMBER

N/A

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)

EOARD
PSC 802 BOX 14
FPO 09499-0200

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

CSP 00-5041

11. SUPPLEMENTARY NOTES

12a. DISTRIBUTION/AVAILABILITY STATEMENT

Approved for public release; distribution is unlimited.

12b. DISTRIBUTION CODE

A

13. ABSTRACT (Maximum 200 words)

The Final Proceedings for Organized Vortical Motion as a basis for Boundary-Layer Control, 20 September 2000 - 22 September 2000

This is an interdisciplinary conference. Topics include 1) Analogies between laminar-turbulent transition and turbulence development applicable to flow control; 2) Application of stability theory approaches to turbulence investigation and control; 3) Receptivity of turbulent boundary layers to induced organized motion; 4) Deterministic elements in methods of turbulent boundary-layer description and control.

14. SUBJECT TERMS

EOARD, Boundary Layer Control, Flow Control, Vortex flows, Transition

15. NUMBER OF PAGES

60

16. PRICE CODE

N/A

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19, SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
UNCLASSIFIED	UNCLASSIFIED	UNCLASSIFIED	UL

Organized Vortical Motion as a Basis for Boundary Layer Control

KIEV 2000

INTERNATIONAL WORKSHOP

20 – 22 SEPTEMBER, 2000

Institute of Hydromechanics,
National Academy of Sciences

Kiev, UKRAINE

Sponsors of the Workshop:

European Office of Aerospace Research and Developmnt (EOARD),
European Division of the Office of Naval Research (ONRIFO),
Science and Technology Center in Ukraine (STCU).

The Workshop was supported by the American Institute of Aeronautics and Astronautics (AIAA) and National Academy of Sciences of Ukraine

Organized by

Institute of Hydromechanics, National Academy of Sciences of Ukraine,
Grinchenko, V. *(Director)*
Nikishov, V. *(Deputy Director)*
Voropaev, G. *(Head of the Department)*
Yurchenko, N. *(Co-Chair)*
Koryenna, L., Rozumnyuk, N. *(Scientific Secretaries)*

SCIENTIFIC COMMITTEE

Alekseenko, S. *(Inst. of Thermal Physics, Novosibirsk)*
Artyushin, L. *(Air Force Institute, Kiev)*
Beutner, T. *(AFOSR, USA)*
Cantwell, B. *(Stanford University, CA, USA)*
Criminale, W., ***Co-Chair*** *(University of Washington, WA, U.S.A.)*
Epik, E. *(Institute of Engineering Thermophysics, NASU, Kiev)*
Grinchenko, V. *(Institute of Hydromechanics, NASU, Kiev)*
Hopfinger, E. (*IMG, Grenoble, France)*
Kim, J. *(UCLA, USA)*
Kozlov, V. *(Inst. of Theor. and Appl. Mechanics, Novosibirsk)*
Launder, B. (*UMIST, Manchester, UK*)
Lipatov, I. *(Central Aero-Hydrodynamical Inst., TsAGI, Moscow)*
Liu, J. *(Brown University, RI, USA)*
Luchini, P. (*Milano University, Italy*)
Maurice, M. *(AFOSR, AFRL Internat. Office, USA)*
Nikiforovich, E. *(Institute of Hydromechanics, NASU, Kiev)*
Rivir, R. *(Wright-Patterson AFB, OH, USA)*
Saric, W. *(Arizona State University, AZ, USA)*
Savchenko, Yu. *(Institute of Hydromechanics, NASU, Kiev)*
Stupin, A. *(State University, Donetsk*)
Suzuki, K. (*Kyoto University, Japan)*
Voropaev, G. *(Institute of Hydromechanics, NASU, Kiev)*
Yurchenko, N., ***Co-Chair*** *(Institute of Hydromechanics, NASU, Kiev)*
Zhou, H. (*Tianjin University, China*)

CONTENTS

Dynamics of Large-Scale Structures in an Impinging Jet Flow

[1]S. Alekseenko, A. Bilsky, D. Markovich, V. Vasechkin

In the near field of the free jet shear layer the large-scale vortex structures (LSVS) contain the main part of the turbulent kinetic energy and influence substantially the intensity of mixing and heat transfer. These structures develop initially as instability waves near the nozzle edge, then they roll up into discrete vortices which can further merge with the creation of more large coherent formations. An active control of the flow can be provided by the external excitation which can lead to resonant amplification of large-scale vortices. At some conditions the broad-band turbulence is partially suppressed and this situation can be considered as quasilaminarisation of the flow (Alekseenko, Markovich & Semenov, 1997). The first comprehensive study of free forced axisymmetric jet was done by Crow & Champagne (1971). The number of experimental and theoretical studies exists on the development of large-scale instabilities in the free shear layers and jets and testing the different ways of the active and passive control the turbulent structure. However, as for confined jet flows, the stability analysis was done only by Ho & Nosseir (1981) who supposed, in particular, that the decrease of vortices frequency occurs as a result of collective interaction of large-scale structures in the near-wall region of high speed impinging air jet. Later, only few works have touched some partial effects concerning coherent structures in confined jet flows.

The present work is devoted to the study of instabilities evolution during round jet impingement under the action of low-amplitude periodical forcing. The main results are obtained with the aid of electrodiffusion method for measuring velocity and wall shear stress (Alekseenko & Markovich, 1994) and also by the DANTEC Particle Image Velocimeter (model 2D PIV 1100 with double Nd-YAG laser).

The used working fluids were: the special electrochemical solution for electrodiffusion measurements and distilled water with small seeding particles (of 20÷50 μm diameter) for PIV measurements. The submerged round jet issued from the well-profiled nozzle and impinged normally on the measuring plate (Fig. 1). The skin friction probes were placed at the measuring plate which could be shifted and this allowed for changing the radial position of each probe with an accuracy of 0.1 mm.

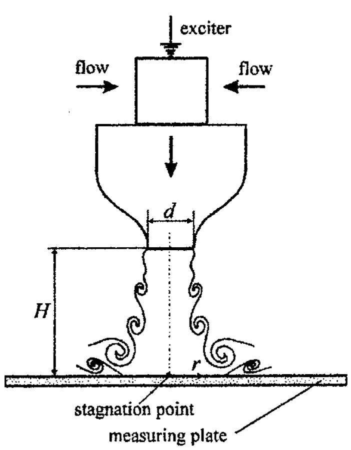


Fig.1. Sketch of impinging jet flow

The experiments were carried out in a wide range of Reynolds numbers Re: $10^3 < Re < 8.2 \cdot 10^4$, where $Re = u_0\, d/\nu$, u_0 is the mean flow rate velocity at the nozzle exit, d is the nozzle diameter. The velocity profile at the nozzle exit was uniform and the turbulence level T_{um} did not exceed a value of 0.005 (in the absence of forcing).

The stability analysis was made on the basis of wall shear stress measurements. Spectral processing of skin friction pulsations was provided by discrete Fourier transform. The spectral analysis was performed for the different frequencies and amplitudes of forcing pulsations. The Strouhal number, Sh, builded on the basis of excitation frequency was varied in the range of 0.2 ÷ 1.0 and the maximum nondimensional level of forcing amplitude was about 0.001. Measurements showed that either enhancement or suppression (Fig.2) of broad-band turbulence in impinging jet can be observed depending of forcing conditions.

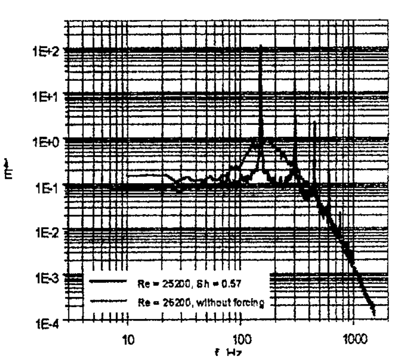
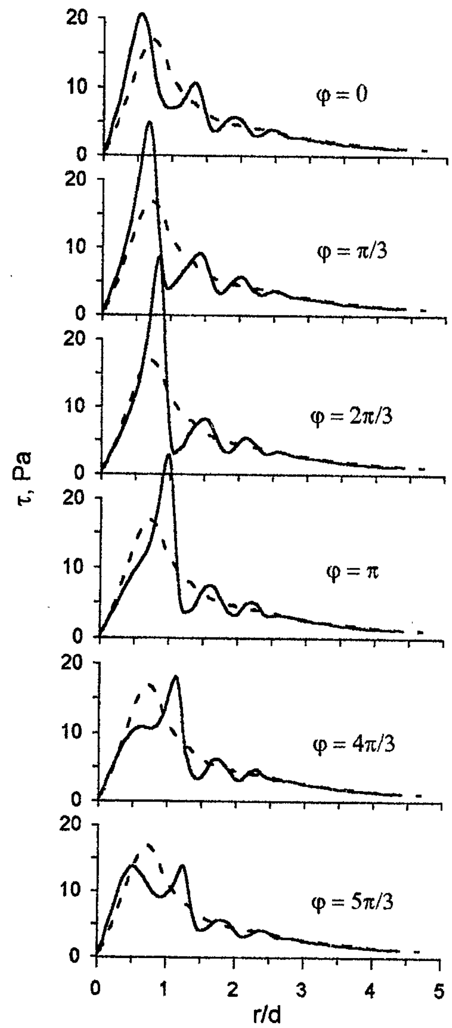


Fig.2. Effect of forcing on the power spectra of wall shear stress in impinging jet. H/d = 2, r/d = 1.0. Level of excitation: ε ≈ 0.0005

The experiments showed also that usually the fundamental harmonic achieves its maximum at the radial distance close to $r/d \approx 1.0$. The maximum of subharmonic is observed at $r/d \approx 2.0$ where the merging of coherent structures penetrating from the free jet shear layer occurs with highest probability. Development of multiple frequences depends on the Reynolds number and forcing conditions.

The technique of conditional sampling (phase averaging) allowed us to measure the instantaneous pattern of the flow, averaging only by broad-

[1] *Institute of Thermophysics, Siberian Branch of RAS, Lavrentyev Ave., 1, Novosibirsk, 630090, Russia; e-mail:* aleks@itp.nsc.ru, *phone:(3832)342050*



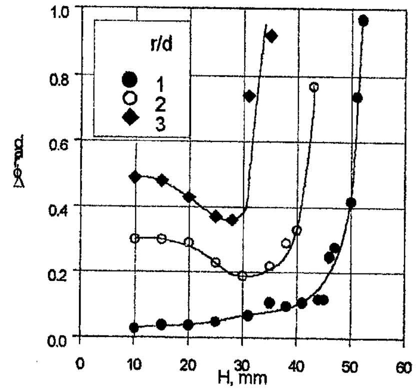


Fig. 4. Phase difference between the coherent pulsations in two symmetrical points on the measurement plate. Re = 25200, Sh = 0.57. Point of symmetry - stagnation point.

Fig. 3. Phase-averaged distributions of wall shear stress. Forced impinging jet. Re = 12700, Sh = 0.53, H/d = 2.

band turbulence. This approach gives the possibility to show the dynamics of structures propagation along stagnation wall. The example of phase-averaged wall shear stress distributions is plotted in the Fig. 3. Each solid curve corresponds to some structure's phase. Dashed line corresponds to the whole-averaged values of wall shear stress. The main conclusion can be made from these experiments - during jet impingement with amplified large-scale structures one can expect very high local values of skin friction and, correspondingly, heat fluxes in non-isothermal systems. Phase-averaged measurements of the velocitiy and vorticity fields were also performed by Particle Image Velocimetry. In particular, the phase velocities of LSVS were obtained for different boundary conditions of the flow. During LSVS propagation along the impingement wall their speed changes by nonmonotonical way - the zones with local slowing-down and acceleration were observed.

With the use of double skin friction probes the local back flows in the near-wall region were observed. Two regions with local flow separations have been found - stagnation point region and small zone with the radial coordinate equals approximately to $r/d = 1.3$, where propagating large-scale structure initiates the local positive pressure gradient.

It is known that in free axisymmetric jet or wake flow the initially axisymmetric vortex structures lose their symmetry downstream and spiral instability modes appear to be most incremental. In the case of impinging jet flow, the phase measurements allowed us to observe the loss of axisymmetric character of the structures with the increasing of their propagation path (from the nozzle exit and up to measurement points at the plate). Fig. 4 demonstrates the growth of phase difference between coherent pulsations both for increasing of nozzle-to-plate distance and radial distance on the plate.

References

1. **Alekseenko, S.V. & Markovich, D.M**. 1994. Electrodiffusion diagnostic of wall shear stresses in impinging jets. *J. Appl. Electrochemistry*, **24**, pp. 626-631.
2. **Alekseenko, S.V., Markovich, D.M. & Semenov, V.I.**, 1997, "Effect of external disturbances on the impinging jet structure," *Proceedings, 4th World Conference on Experimental Heat Transfer, Fluid Mech. and Thermodynamics*, Brussels, June 2-6, 1997, Vol. 3, pp. 1815-1822.
3. **Crow, S.C. & Champagne, F.H**. 1971. Orderly structure in jet turbulence. *J. Fluid Mech*, **48**, pp. 547-591.
4. **Ho, C.M. & Nosseir, N.S.** 1981. Dynamics of an impinging jet. Part I. The feedback phenomenon. *J. Fluid Mech*, **105**, pp. 119-142.

Organized Vortical Structures Induced by System Rotation: Computer Experiments and Mathematical Modelling

Helge I. Andersson[1]

Background

The presence of organized vortical structures in wall-bounded flows are known to occur both in flows affected by centrifugal forces due to streamline curvature and by Coriolis forces due to imposed system rotation. Such flows, be it boundary layers, channel or duct flows, are susceptible to a roll-cell instability analogous to the Taylor-instability in a flow between independently rotating cylinders. The development of a spanwise array of vortical roll cells of alternate sign aligned with the primary flow stems from a centrifugal or Coriolis instability mechanism, and may occur irrespective of whether the flow is laminar or turbulent; see e.g. Andersson (1999).

With the view to provide further insight into rotational-induced counter-rotating vortical structures or roll cells, statistically fully developed Couette flow between two infinite parallel planes have been considered. The flow is rotated about a spanwise axis, i.e. in orthogonal mode, so that the rotation vector is either parallel or antiparallel both to the walls and the mean vorticity vector. The absence of side-walls eleminated the possible occurrence of other kinds of secondary motions normally induced by lateral boundaries. The rationale for considering shear-driven Couette flow rather than pressure-driven Poiseuille flow is that the former is exposed entirely either to cyclonic or anti-cyclonic rotation. Here, the focus will be on anti-cyclonic rotation, i.e. the imposed rotation (or background vorticity) remains parallel with the vorticity associated with the primary fluid motion throughout the entire flow field.

Computer Experiments

The unsteady Navier-Stokes equations governing the turbulent motion of an incompressible fluid are represented on a three-dimensional computational mesh and integrated numerically without any further assumptions than those involved in the replacement of the governing partial differential equations by non-linear discretized momentum equations. The directly simulated flow fields can thus be considered as numerical realizations of real turbulence, provided that the resolution in time and space is of the order of the tiniest scales of the turbulence.

Such computer experiments have been carried out for plane Couette flow with a Reynolds number equal to 5200 (based on the velocity U_w of the moving wall and the distance H between the walls. The relative importance of the Coriolis force and inertia is conventionally expressed in terms of a rotation number $Ro = 2\Omega H/U_w$ in industrial fluid dynamics and as a Rossby number equal to $1/Ro$ in geophysical and astrophysical fluid dynamics. After first having verified the computer simulations of non-rotating Couette flow, i.e. $Ro = 0$, flows at a number of different rotation numbers were examined, ranging from weak via intermediate to strong rotation, see Bech & Andersson (1996, 1997). Even at $Ro = 0.01$, a persistent pattern of counter-rotating vortices could be observed and a triple-decomposition of the instantaneous velocity field was devised in order to distinguish between the three-componential mean flow field associated with the vortical roll cells and the underlying 3D turbulence. It could then be shown that the turbulence structure was practically unaffected by the presence of the roll cells at this low rotation, but more surprising was the observation that the level of the kinetic energy of the turbulence was reduced in comparison with the non-rotating flow. At the intermediate rotation numbers 0.10 and 0.20, the vortical roll cells were far more energetic than at $Ro = 0$ and contained four times the energy contained in the real turbulence. At $Ro = 0.50$, a roll-cell breakdown was observed and only reminiscences of the vortical structures were left, while a substantial increase in the energy content of the turbulence took place, even exceeding the turbulence level in the non-rotating case. This phenomenon was accompanied by a substantial amplification of the streamwise vorticity fluctuations.

[1] Division of Applied Mechanics
Norwegian University of Science and Technology
N-7491 Trondheim, Norway
E-mail: Helge.I.Andersson@mtf.ntnu.no

Mathematical Modelling

Computational flow analysis will in the foreseeable future be based on the Reynolds-averaged Navier-Stokes (RANS) equations in combination with a suitable turbulence model to account for the *a priori* unknown Reynolds stresses. The turbulence model is formally intended to account only for the real turbulence, but numerous examples of RANS-based computations have failed to distinguish between organized vortical structures induced by a destabilizing body force and the turbulent velocity fluctuations; see e.g. Pettersson & Andersson (1997). Admittedly, however, the distinction between the organized flow structures and the turbulence cannot be made unless the roll-cell pattern is persistent. Therefore the data base deduced from the computer experiments described in the preceding section represents a unique test case for advanced turbulence models.

In a recent investigation by Pettersson Reif & Andersson (2000), the ability of a full second-moment closure (SMC) to reproduce the rotational-induced flow phenomena observed in the computer experiments was considered. For the first time in a full SMC study, the vortical roll cells caused by the Coriolis force were treated as an integral part of the three-componential mean flow governed by the RANS-equations and the SMC was therefore assumed to account for nothing but real turbulence. For this purpose we used an approach in which near-wall effects are accounted for by elliptic relaxation and the common use of non-linear wall-damping functions could be avoided. System rotation was included naturally in the rotational stress-producing terms and in the mean intrinsic vorticity in the non-linear variable-coefficient pressure-strain model.

Without any attempts to sensitize the model equations for rotation, the model predictions mimicked the most striking effects of anti-cyclonic rotation, as observed in the computer experiments. In particular, they reproduced the characteristic energetic vortical roll-cell pattern, which inevitably enhanced the cross-sectional mixing. The broadening of the central core region, in which the primary mean velocity profile adjusted itself to make the absolute mean vorticity negligibly small, was well captured by the predictions, together with the remarkable damping of the turbulent velocity fluctuations. The present results support our view that the need to modify a SMC model is reduced or even eliminated if the mean flow is properly resolved.

Acknowledgements

Substantial contributions have been made by my former Dr.Ing. students K.H. Bech and B.A. Pettersson Reif. Computer time was supported by the Research Council of Norway.

References

Andersson, H.I. (1999) Organized structures in rotating channel flow. In: *Simulation and Identification of Organized Structures in Flows*, Kluwer, pp. 81-90.

Bech, K.H. & Andersson, H.I. (1996) Secondary flow in weakly rotating turbulent plane Couette flow. *J. Fluid Mech.* **317**, 195-214.

Bech, K.H. & Andersson, H.I. (1997) Turbulent plane Couette flow subject to strong system rotation. *J. Fluid Mech.* **347**, 289-314.

Pettersson, B.A. & Andersson, H.I. (1997) Near-wall Reynolds-stress modelling in noninertial frames of reference, *Fluid Dyn. Res.* **19**, 251-276.

Pettersson Reif, B.A. & Andersson, H.I. (2000) Prediction of longitudinal roll cells in rotating plane Couette flow. *Theoret. Comput. Fluid Dynamics* (in print).

About a Lift Coefficient of a Ship Rudder with Protectors

V.V.Babiy[1], V.V.Moroz[2]

The ship rudder is the relevant member of a powerhelm unit [1, 3]. The total number of rudder modifications includes many tens.

In practice of shipbuilding the enough large data volume used for hydrodynamic calculation of rudder is accumulated [4]. As a rule, these data have trial-and-error or semiempirical nature and are founded on structural relations of a finit-span wing theory.

General requirement by selection of rudder members is the necessity of maintenance of maximum lift on a control surface in all range of rudder angles from 0 up to 32 ... 35 degrees at minimum values of a torque on a rudder head. This requirement as a matter of fact is rather contradictory: on the one hand - desire to receive high performance of a control surface on small rudder angles constrains to apply rudder of enough large span-chord ratio (1,5 ... 2), and on the other hand - the rudder with such aspect-ratio have a small critical angle of attack (angle of attack, at which one the flow separation starts). For example, the wing of a NACA0024 profile by aspect-ratio 2 has a critical angle of attack about 25 degrees, that at rudder angle in 35 degrees is completely unsatisfactory. The reduction of span-chord ratio leads in a decrease of steepness of dependence of a lift coefficient from an angle of attack, and by that - to a decrease of rudder efficiency on small rudder angles. The greatest expansion rudder of large aspect ratio have received on tow vessels and trawler ships, which one, as a rule, have a design trim by the stern.

For protection of designs of a powerhelm unit against corrosion in practice of shipbuilding the protection will widely be used, the members by which one are set as well on a ship rudder. From the hydrodynamic point of view the protectors represent local vortex generators, the vortex wake from which one is diffused streamwise chord-wise of wing. Influencing this wake on rudder lift as a whole is of interest.

In the report the outcomes of experimental researches of hydrodynamic parameters of model of a ship control complex in a hydrodynamic tunnel are set up [2]. The ship control complex including balanced rudder with a NACA0024 profile, aspect ratio 2,2 and factor of compensation 0,25 was simulated. Besides on a control surface the protection including 6 pair protectors such as P-KOA-10 is established, the installation diagram which one is shown in a fig.1. In rudder model pursuant to the installation diagram of protection the members of attachment of protectors simulators were stipulated. Besides, in a plane of the installation of an extreme lower protector on a suction surface of rudder model the drainage system from openings a diameter of 1 mm and step of 10 mm was made, which one allowed to execute of flow visualization along a rudder profile by painting of water.

The water velocity in a hydrodynamic tunnel during realization of experiment was set on a mark 2 m/s, that at temperature of water 14°C has compounded Reynold's number $1.7\cdot10^5$, computed on a midchord.

As a result of serials of experiments the graphic dependences of hydrodynamic forces coefficients to a rudder angle were plotted. Thus by the begining the rudder model without established protectors, and then with protectors was tested.

As an effectiveness criterion of a control surface it is accepted a mean of a derivative of a lift coefficient on a rudder angle ($Cy/\partial\delta$) in up to critical range of rudder angles.

As the influencing of protection elements on a lift coefficient is visible from a fig. 2 was exhibited very strongly. Strong is watched (up to 25 %) decrease of efficiency of a steerer in range of rudder angles from 0° up to 25°. The conducted of a flow visualization pattern has shown, that behind of protection elements the vortex wake will be derivated, which one displacing lengthways of control surface on flow divides it spanwise conditionally into 4 parts.

[1] *Central designer bureau "Schuna", Kiev*
[2] *Institute of Hydromechanics NAS of Ukraine, Kiev*

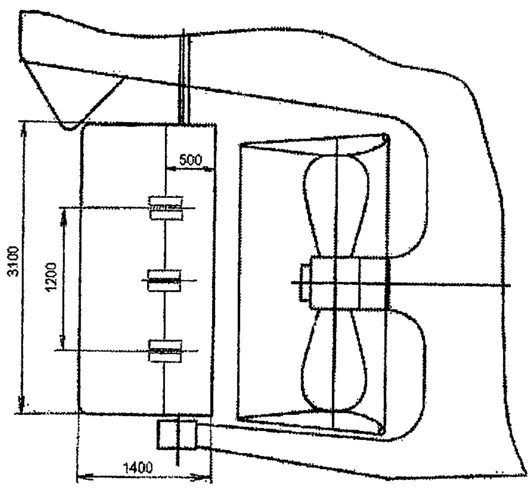


Fig. 1. The scheme of base version of a rudder complex.

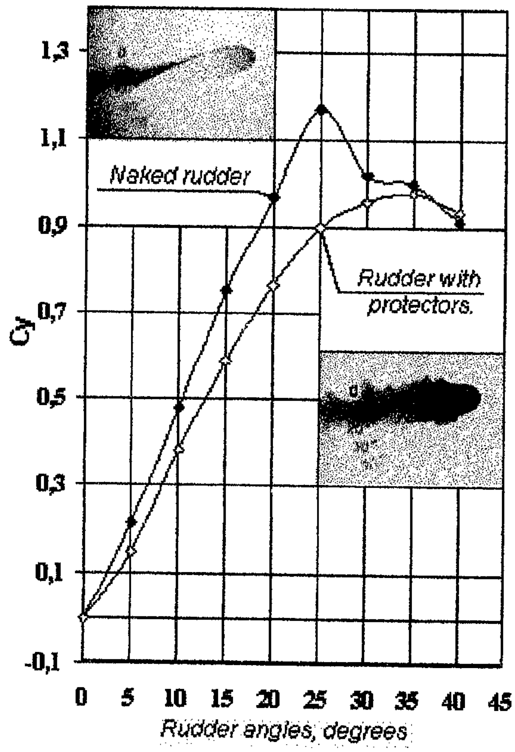


Fig. 2. Dependence of a lift coefficient of to a rudder angle.

Since the vortex wake behind of protection elements is arranged in immediate proximity to a surface of a rudder, it interacts with a boundary layer and exerts influence on pressure profile on a surface of all steerer. It is integrally this influencing it is possible to estimate as decreasing of effective lengthening of a rudder. Thus, the critical angle of attack of a rudder with established protection elements has compounded 35° against 25° for a naked control surface. The value of a derivative of a lift coefficient on a rudder angle $\partial Cy/\partial\delta$ for a rudder with established protection elements has compounded 2,2 rads^{-1}, while for a naked control surface $\partial Cy/\partial\delta = 2{,}77$ rads^{-1}, that is a little bit lower, than for a flap-type rudder of the same aspect ratio.

1. **Hofman A.D.** A powerhelm unit and manoeuvring of a ship. The reference book. - *Leningrad: Shipbuilding, 1988. - 360 p, ill.*
2. **Kayan V.P., Pyateckiy V.Y.** The biohydrodynamic installation of a selfcontained type for research of hydromechanics of sailing marine animal swimming// *Bionics, №5, 1971, pp. 121-125.*
3. **Katsman F.M., Kudrevatiy G.M.** Designing of propeller-rudder complexes of ocean ships. - *Leningrad: Shipbuilding, 1963. - 510 pp, ill.*
4. The reference book on a ship theory: In three volumes. - *Leningrad: S. The Thom 3. A controllability displacement ships. Hydrodynamics of ships with dynamic principles of maintenance / Under edition Y.I. Voytkunskiy hipbuilding, 1981*

Visualization of Vortical Flow and Separation about a Rectangular Bluff Body

D. Banks[1], R. N. Meroney[2]

Introduction

Wind tunnel and full scale pressure studies of flow over rectangular bluff bodies have repeatedly shown that the largest mean, peak, and RMS suction values are observed for taps beneath the separated flow. The precise manner in which the suction depends upon recirculating flow behaviour has not been established. In addition, it has been repeatedly noted that for wind tunnel tests, the peak and RMS pressure coefficients are consistently and appreciably lower than those seen at larger scales, in particular under the recirculation. There has been some consideration of which upstream flow parameters (especially turbulence parameters) govern the nature of the flow within the separation regions, and thus control the nature of the suction, particularly the peaks. However, "no comprehensive experimental results are available for the variations of surface pressures (on roofs) with the nature of the incident flow" (Tieleman et al, 1994). To better understand the flow mechanism in the recirculation which producing these negative pressure coefficients, and to thus gain some insight into how the upstream flow influences the upper surface pressure coefficients Colorado State University has undertaken a model-scale visualization study of the flow in the separation and recirculation zones, under the auspices of the NSF Cooperative Program in Wind Engineering (CPWE).

Method

The initial phase of this study was performed in the 365 cm wide by 200 cm tall CSU Environmental Wind Tunnel (EWT). A 60 cm x 80 cm rectangular wing with skirt lengths between 15 cm and 60 cm was mounted out of the boundary layer, at a height of 100 cm. No roughness elements or spires were in place, so the flow was essentially smooth. Glycerin smoke was introduced through holes near the wing's leading edge for visualization. The visualization plane was illuminated with a laser light sheet produced by a Coherent Innova 70-5 argon-ion water- cooled laser operating in multi-line mode. The laser has a nominal power of 5 Watts, but was generally run at 1 W for these tests. The laser beam was reflected and spread into a sheet by lenses mounted on the tunnel ceiling. Images were recorded on an SVHS camera.

The visualization equipment were moved to the CSU Industrial Wind Tunnel (IWT), where a smaller aluminum square wing model was used, since the test section measures 180 cm x 180 cm. The IWT provides a higher maximum speed, which enhanced the pressure measurements to be taken in the next phase of the experiment. In the second phase, pressures (including pressure peaks) were correlated with images simultaneously recorded at 60 Hz.

Some of the difficulties in extracting quantitative information from the SVHS recordings have been addressed through the implementation of a digital recording system. This system makes use of a Pulnix TM-7CN CCD array camera. This camera provides a high resolution (768 x 494 pixels) grey scale image, and has a high sensitivity in the range of the principal wavelengths of the argon-ion laser (0.5 Lux at F=1.4). It also has a variable shutter speed (1 /60^{th} sec to 1/10^4 sec), permitting better temporal resolution of flow behaviour. The signal from the camera is fed to an Imaging Technology Inc MVC IC Image Capture Board, placed in a Pentium PC with a PCI bus. The image processing (IP) board comes with an IP library (written in c/c++), and software has been written to acquire single images, averaged images, and image sequences of up to 6 sec in length. These images are then viewed and processed digitally (measured, averaged, thresholded, subtracted, etc).

Conclusions

While the prevalence of high suctions in the separated flow regions has been noted for all directions of the approaching wind, the worst suctions of all are associated with the wind angles between 60° and 45° (Lin et al, 1995). It is known that at these angles, the flow separation induced

[1,2] Fluid Mechanics and Wind Engineering Program, Civil Engineering Department, Colorado State University, Fort Collins, Colorado, 80523

by the roof's two "leading edges" contains a pair of conical vortices, named "delta wing" vortices, as they resemble the vortices seen above the wings of delta wing aircraft. It is generally acknowledged that these vortices are responsible for the high suctions, which increase exponentially toward the roof corner (Lin et al, 1995). Once again, though, "the mechanism linking vortex structure and surface pressure is little understood" (Marwood and Wood, 1996).

The first issue addressed was the location and size of the vortices; where is the core, and how is its position (height above the roof and distance from the roof edge) affected by the wind angle (ω), wind speed, or other flow conditions. It was found that the mean vortex core position varied linearly with distance from the apex, and was independent of wind speed for the range of speeds tested (1 m/s to 10 m/s).

It has been accepted that the peak mean suctions lie along a line beneath the mean vortex position. This line or ray's position is often described by the angle that this line forms with the roof edge (ϕ). Results for both the delta wing and for various low rise building models indicate that ϕ varies from 5° to 30° as the wind angle in increased from 0° (parallel to the wall) to 70° (nearly perpendicular to the wall). Both turbulence intensity and wall length are expected to affect Cp and $\phi(\omega)$ (see Kawaii and Nishimura, 1996).

Subsequent work has focussed on the correlation of vortex behaviour and surface pressures. Image information gathered concerns variations in vortex size and location as a function of wind angle and flow conditions. For example, the data indicates that the greatest suctions do not occur for the most steady vortex, but rather for wind angles near 55°, which provide a less predictable vortex position, while still providing enough of an angle for the vortex to consistently appear. (The vortex becomes more intermittent and difficult to discern for ω much above 65°, but structures are still visible in the flow as the recirculating flow makes the transition from conical vortex to bubble separation.) An analytic model was created to explain the flow physics of the intermittent high suction speeds, which has been validated by comparison with experimental data.

The full paper will present additional mean and intermittent vortex geometry information from the model scale tests. The correlation of local pressures and fluid kinematic perturbations seen with simultaneous visualization and velocity measurements will be emphasized.

References

- D. Banks, The Suction Induced by Conical Vortices on Low-Rise Buildings with Flat Roofs, Ph.D. dissertation, Colorado State University, Civil Engineering Department (2000)
- D. Banks and R.N. Meroney, A Model of Roof-Top Surface Pressures Produced by Conical Vortices: 1) Model Development, 2) Evaluation and Implications, accepted by J. Wind and Structures, (2000)
- D. Banks and R.N. Meroney, The Applicability of Quasi-Steady Theory to Pressure Statistics Beneath Roof-top Vortices, accepted by J. Wind Eng. Ind. Aerodyn, (2000)
- D. Banks, R.N. Meroney, P.P. Sarkar, Z. Zhao, and F. Wu, Flow visualization of conical vortices on flat roofs with simultaneous surface pressure measurement, J. Wind Eng. Ind. Aerodyn., Vol. 84(1) (2000), pp. 65-85.
- L. S. Cochran, Wind Tunnel Modelling of Low Rise Structures, Ph.D. dissertation, Colorado State University, Civil Engineering Department. (1992)
- H. Kawai and G. Nishimura, Characteristics of fluctuating suction and conical vortices on a flat roof in oblique flow, J. Wind Eng. Ind. Aerodyn., 60 (1996) 211-225
- J.-X. Lin, D. Surry, and H.W. Tieleman, The distribution of pressure near roof corners of flat roof buildings, J. Wind Eng. Ind. Aerodyn., 56 (1995) 235-265
- R. Marwood and C. J. Wood, Conical vortex movement and its effect on roof pressures, presented at the Bluff Body Aerodynamics and Applications Colloquium, Virginia, USA, (1996).
- H. Okada and Y.-C. Ha, Comparison of Wind tunnel and Full-scale pressure measurement tests on the Texas Tech building, J. Wind Eng. Ind. Aerodyn., 41-44 (1992) 1601-1612
- H. W. Tieleman, Model/full scale comparison of pressures on the roof of the TTU experimental building, Proceedings of the East European Conference on Wind Engineering, Warsaw, Poland (1994)
- H. W. Tieleman, D. Surry, and J.-X. Lin, Characteristics of mean and fluctuating pressure coefficients under corner (delta wing) vortices, J. Wind Eng. Ind. Aerodyn. 52 (1994) 263-275.

Improvement of the Aerodynamic Characteristics of Lifting Surfaces of Transport Aircraft by Methods of the Boundary Layer Management

Oleg K. Bogdanov[1]

ANTONOV ASTC jointly with ZAGI execute the experimental and computational target researches on models of transport aircraft, directed on obtaining of the traditionally high aerodynamic characteristics and ensure of reliability and safety of flights.

There are submitted same results of this researches in part of:
- further perfecting of the ways and means of the wing lift increase (deflection of flaps on large angles, blowing of wing by streams from propeller and reactive jets, management of the boundary layer on the flaps and control surfaces by a way of blowing of the compressed air);
- integrated improvement of local aerodynamics of aircraft on models and elements of models with the purpose of the profile resistance reduction, including use of laminar profiles and management of the laminar flow by a way of suction of the boundary layer on compartment of wing;
- increase of effectiveness of the control surfaces and reduction of the hinge moments by a way of management of the boundary layer on control surfaces and ailerons;
- ensure of the longitudinal static stability of aircraft in all range of flight angles of attack and Mach numbers with application of vortexes on a wing.

Results of the conducted researches are introduced by development and creation of ANTONOV ASTC An-124, An-72, An-38, An-70, An-140 transport aircraft.

[1] Antonov Design Bureau, Kyiv, Ukraine

About Interaction of a Running Pressure Perturbation with a Boundary Layer

Vladimir Bogolepov[1], Vladimir Neyland[2]

The flow of the viscous perfect gas over a surface is considered. It is supposed that the Mach number in the uniform undisturbed stream is supersonic $(M_\infty^2 - 1) \sim 1$, and the Reynolds number is large but subcritical $Re_\infty = \rho_\infty u_\infty l / \mu_\infty = \varepsilon^{-2} >> 1$. Here ρ_∞, u_∞ and μ_∞ are the gas density, velocity and viscosity coefficient in the uniform undisturbed stream and l is some distance from the surface leading edge. The disturbed local region at the distance l from the surface leading edge is investigated which induced by the running along a surface pressure perturbation. We will use common nondimentional variables only. It is supposed that the pressure perturbation and its running speed are small $\Delta p \sim \theta << 1$, $u_{\Delta p} \sim \sigma << 1$.

If the pressure perturbation induces in the boundary layer near-wall part, where the gas velocity is small $u \sim u_{\Delta p} \sim \sigma << 1$ and viscosity effects are insignificant, a nonlinear velocity disturbances $\Delta u \sim u$, then the pressure perturbation value is $\Delta p \sim \theta \sim \sigma^2 >> \varepsilon^{1/2}$. Supposing that the pressure perturbation is self-induced, it is possible to construct the self-sustained system of estimations for the tripe-deck flow. In the coordinates system running together with the pressure perturbation with constant speed $u_{\Delta p} = \sigma c$, $c \sim 1$ the analytical expression for gas velocity near the wall is obtained

$$U = \frac{1 \pm \exp(X + b)}{1 \mp \exp(X + b)} \qquad (1)$$

Here the longitudinal variable X is set to within any constant b, the top sign concerns to branches of the solution at $|U| \geq 1$ and the bottom sign concerns to branches of the solution at $|U| \leq 1$.

It is shown that at $U, c > 0$ when the pressure perturbation goes towards to the basic flow, the subcritical flow realizes and the solution (1) describes the upstream influence. When $U,\ c < 0$ and the pressure perturbation goes in a direction of the basic stream, the supercritical flow realizes and the solution (1) describes the downstream decreasing.

It is marked, that here the boundary layer displacement thickness change is created by the variation of the near-wall nonlinear region thickness.

The developed theory of interaction a running pressure perturbation with a boundary layer on a flat surface allows to expand a class of researched flows and to consider, for example, the case of a flow near a surface with a fracture.

When the pressure perturbation $\Delta p \sim \theta << \sigma^2$ induces only linear inviscid disturbances the boundary layer thickness change is $A \approx \exp(\pm X)$. This solution describes disturbances increasing at $X > -\infty$, $c > 0$ for the subcritical flow (the top sign) or disturbances decreasing at $X < \infty$, $c < 0$ for the supercritical flow (the bottom sign). In this case the change of the boundary layer displacement thickness is created by the variation of the near-wall linear region thickness but not the nonlinear critical layer.

When the inviscid linear near-wall layer is non-stationary the algebraically dispersion relation is obtained.

This work is supported by the Russian Foundation for Basic Researches (projects code are 98-01-00462 and 00-15-96070).

[1,2] *Central Aerohydrodynamic Institute (TsAGI)*
Zhukovsky-3, Moscow Region, 140160, Russia
Phone: (7 095) 556-49-87; Fax: (7 095) 911-00-19; E-mail: bogol@tsagi.rssi.ru

Experimental Simulation of Sweapt Wing Boundary Layer Receptivity to Vortical Disturbances at High Free Stream Turbulent Level

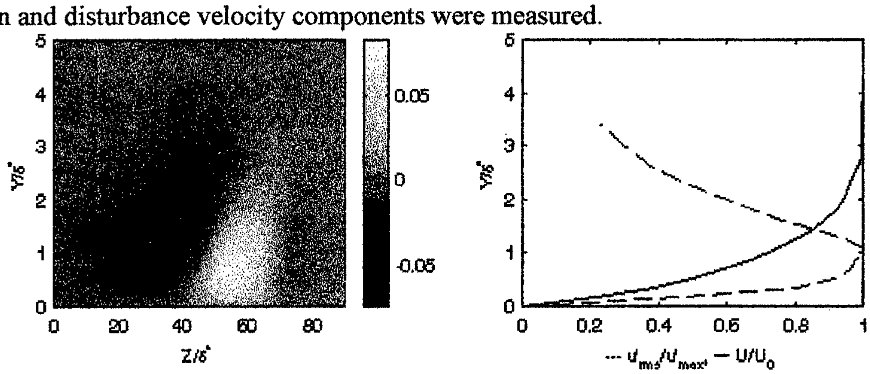
A.V.Boiko [1]

Pretransitional development of two- and three-dimensional boundary layers subjected to nearly isotropic free-stream turbulence (FST) has been investigated in a series of studies [1, 2, 3]. A direct validation of a theoretical model for receptivity of Blasius boundary layer to FST was previously successfully carried out by an experimental simulation technique [2, 3]. It was proved that excited controlled velocity modulations exhibit the main characteristics of the so-called Klebanoff mode, usually found under effect of 'natural' FST [1]. A theoretical hypothesis, that flat plate leading edge plays no significant role in the receptivity process to the vorticity, produced by the micro-wing tip [2] was confirmed in [3].

The main aim of present work is to provide solid experimental basis on receptivity of canonical three-dimensional flows to prototypic free-stream disturbances and to give hints for building theoretical and numerical models of the receptivity to FST as well as to obtain a general view about the downstream development of excited boundary layer structures.

Transition in open flows usually depends strongly on initial conditions (i.e. the disturbance field and the flow geometry). It leads to the necessity of a special, so called receptivity, problem which solution must connect appearance of boundary layer disturbances with external perturbations (e.g., FST). Meanwhile, transition process bypasses the stage of instability wave amplification in many real cases through the growth and later breakdown of transient streak-like quasi-stationary disturbances, which specific characteristics strongly depend on causing them surface roughness or FST. The effect of FST on the onset of transition is also of great interest both in applied engineering, e.g., for the prediction of transition on turbine blades affected by turbulence from a stator and in the influence of FST on wind tunnel experiments.

However, the complexity of the initial turbulent free-stream flow-field necessitates a use of experimental simulation technique to consider the receptivity process under controlled conditions, so that the results will be suitable for direct comparison with available and new theoretical and numerical models as well as experiments carried out under 'natural' conditions. For this purpose a vortex originated at the tip of a micro-wing positioned in front of a swept wing model outside the boundary layer was used. By varying the wing angle of attack, the vortex strength was controlled. To produce low-frequency vortex modulations, the angle of attack could be varied periodically by a special device. At first, to provide initial conditions for the vortex-boundary layer interaction, the tip vortex axial and circumferential velocity components were measured in several planes downstream the micro-wing with the help of hot-wire anemometry. Also, a supplementary hydrogen-bubble visualisation in a water tunnel was done. Then, the boundary layer response dominated by large scale narrow structures growing downstream by amplitude was investigated in the same downstream planes. Stream- and span-wise mean and disturbance velocity components were measured.



[1] *Institute of Theoretical and Applied Mechanics, Novosibirsk 630090, Russia*†
Fax: 007 3832 342268 e-mail: boiko@itam.nsc.ru

A sample of iso-colour distributions of stream-wise velocity defect induced inside the swept-wing boundary layer (left); normalised undisturbed mean velocity profile (U/U_0) and averaged velocity defect profile (u'_{rms}/u'_{max}) across the boundary layer (right).

The hot-wire measurements and visualisation data demonstrate the tip vortex behaviour and the wake behind the micro-wing in the free stream: laminarity of the flow, symmetry of the vortex, presence of axial and azimuthal vortex velocities. Mean and time-dependent characteristics of the induced boundary layer structures in the flows under consideration were documented: displacement and momentum loss thicknesses as well as shape factor modifications produced by the vortices of different intensity and their downstream behaviour; the shapes of the stationary vortices and their low-frequency oscillations at different downstream positions; the vortex downstream growth rates (based on different criteria).

The formation of the vortical boundary layer structures over the swept wing, in contrast to 2D boundary layer [3], starts close to the nose that is due to this principal change of the initial conditions and stability characteristics to stationary disturbances. A separate investigation of stationary spatial waves constituting the vortices was performed: a spatial spectral transformation of the experimental data via Fourier integral was applied and behaviour of the growth of the power density of the stationary harmonics constituting the vortex was considered.

Under effect of the FST, the quasi-stationary streaks cannot lead to turbulence directly, rather it seems they are subjected to a high-frequency instability. In the cases under consideration it seems reasonable to introduce controlled disturbances inside the vortices and investigate their development. Such instability studies are now in preparation. Their general goal is to deepen our knowledge on the mechanisms of generation, development and interaction of pre-transitional disturbances of different types as well as development of methods of description of such structures and methods of their effective control.

The work was supported by Alexander von Humboldt Foundation, Germany and DLR-Institut für Strömungsmechanik, Göttingen, Germany.

References

1. **K. J. A. Westin, A. V. Boiko, B. G. B. Klingmann, V. V. Kozlov, P. H. Alfredsson**, Experiments in a boundary layer subjected to free-stream turbulence. *J. FluidMech., 1994, v. 281, 193-218.*

2. **F. P. Bertolotti, J. M. Kendall**, Response of the Blasius boundary layer to controlled free-stream vortices of axial form. *1997, AIAA Paper, 97-2018.*

3. **A.V.Boiko**. Flat plate boundary layer response to controlled free-stream axial vortices, *Book of Extended Abstracts of 11th International Couette-Taylor Workshop, July 20-23, 1999, Bremen, Germany.*

Heat Transfer under Persistent Vortices

Robert E. Breidenthal[1]

INTRODUCTION

Traditional strategies to controlling wall fluxes have included riblets, large eddy break-up (LEBU) devices, and micro-electronic mechanisms (MEMS). Riblets are designed to affect the smallest eddies. LEBU devices attempt to destroy the largest eddies. MEMS use active feedback to cancel transport.

A fundamentally different approach to reduce wall fluxes has recently been proposed.[1] Instead of modifying the smallest eddies or neutralizing the largest, a strong vortex is deliberated added into a turbulent flow. This is counter-intuitive, since such vortices have long been used to increase the transport within a boundary layer. For example, vortex generators are common on aircraft to inhibit boundary layer separation by increasing the transport of high momentum fluid towards the surface.

However, the present approach exploits a special kind of vortex, one that is stationary, or persistent. According to a new theory originally developed for stratified entrainment,[2] such a persistent vortex would lower the wall flux from a turbulent to a laminar value. That is to say, the wall flux depends on only the square root of the ratio of the fluid diffusivity of the flux in question to the rotation period of the largest eddy, the persistent vortex. This is in contrast with the ordinary turbulent boundary layer, where the flux depends on the rotation period of the smallest eddy, the Kolmogorov microscale. Thus at large Reynolds number, the wall flux for the persistent case would be reduced by a factor of $Re^{1/4}$ from the nonpersistent case according to the theory.

For the imposed vortex to dominate the physics, it must be sufficiently strong compared to the background turbulence. The theory predicts that the induced velocity of the imposed vortex must be greater than the velocity at the Kolmogorov microscale to lower the wall flux.

The imposed vortex must also be sufficiently stationary. Defined to be pi times the ratio of the rotational to the translational speed of a vortex with respect to a nearby interface, the persistence parameter is a measure of the vortex stationarity.

In order to achieve sufficient stationarity without the complexity of active feedback, the vortex is stabilized using a wavy wall as suggested by Balle.[3] In analogy with the von Karman wake, where the vortices are quasi-stable, the wavy wall replaces the dividing streamline of the wake to stabilize the imposed vortices.

EXPERIMENT

A wavy plate was mounted in a water tunnel, with the grooves approximately aligned in the flow direction. Vortex generators were positioned near the leading edge, so that an array of co-rotating vortices is formed, with one vortex in each groove. Experiments confirmed the enhanced stability of the vortices due to the grooves. Correctly positioned vortices are remarkably stationary. However, if the vortex generators are slightly moved, so that the vortices are not correctly positioned, the stability vanishes.

An electric heater on the bottom of the plate generated a heat flux through the plate, measured by stacked thermocouples on each side of the plate. The dimensionless heat flux, the Nusselt number, was measured at the bottom of a groove over the Reynolds number range of 10^4 - 10^5 based on x.

When the vortex generators were positioned so as to form persistent vortices, the Nusselt number was proportional to $Re^{0.58}$, close to the laminar value of $Re^{1/2}$. For the nonpersistent case, the Nusselt number went as $Re^{0.83}$, close to the value for the ordinary turbulent boundary layer at this Reynolds number, $Re^{0.8}$, as expected. The ordinary turbulent boundary layer is nonpersistent, as the strongest vortices in it move rapidly with respect to the plate. The ratio of the Nusselt numbers in the two cases was approximately $Re^{1/4}$, also predicted by theory.

---

[1] *Department of Aeronautics and Astronautics, University of Washington, Seattle, Washington 98195-2400* USA

DISCUSSION

For wall fluxes, the theory asserts that there are two fundamentally different types of turbulence. In nonpersistent turbulence, the wall fluxes are turbulent, in that they depend on the fine scale turbulence. In persistent turbulence, the wall fluxes are laminar, completely independent of the fine scale turbulence. Another example of a turbulent flow yielding a laminar flux is the laminar growth rate of a tip vortex from a lifting wing at large Reynolds number.[4] Rotation induces an effective stratification at radii near the maximum azimuthal velocity. The entrainment rate at such a highly stratified interface around a persistent vortex is the laminar value, independent of the fine scale turbulence.

The obvious question of skin friction drag reduction remains open. While Reynolds analogy predicts that the skin friction would also be reduced, the net drag of generating the vortices must also be included. Although a small fraction of original energy in the vortex may be recoverable at the trailing edge in some applications, the main issue hinges on the unanswered question of how far downstream of the vortex generators the vortices remain persistent.

CONCLUSIONS

The wall heat flux of a turbulent flow was reduced to that of a laminar flow by adding vortices. The flux declined by a factor of approximately $Re^{1/4}$, in accord with persistence theory. This is the first demonstrated control of wall flux by inducing a turbulent flow to yield a laminar flux.

Vortex stationarity was achieved by passive means, without active feedback control. As this technique requires only a specially shaped wall and carefully positioned vortices, it may prove useful in practical applications at large Reynolds number.

References

[1]A.J. Cotel & R.E. Breidenthal 1998 A new approach to drag reduction, *Proc. Int. Sym. Seawater Drag Reduction*, 22-23 July 1998, Newport, Rhode Island.

[2]A.J. Cotel & R.E. Breidenthal 1997 Persistence effects in stratified entrainment, *Applied Scientific Research* **57** 349-366.

[3]G.J. Balle, T.M. Kier, H. Touel & R.E. Breidenthal 1999 Stationary vortices and persistent turbulence in Karman grooves, submitted to *J. Fluid Mech.*

[4]A.J. Cotel & R.E. Breidenthal 1999 Turbulence inside a vortex, *Phys. Fluids* **11**(10) 3026-3029.

An Initial-Value Approach to Flow Control

William O. Criminale[1]

The science of linear analysis is well founded and understood for a wide variety of coupled dynamical systems. And, regardless of the specific dynamical problem that is under scrutiny, it is clear that one must have knowledge of all the dynamics in order to control any deviations that may ensue due to the particular configuration. Even major nonlinear coupled systems are analyzed and configured in this manner. Thus, it is the mathematics that deals with feedback that is used to suppress any detrimental output of a system. Suppression can sometimes be done for all time and, at other times, only delay of the inevitable is possible but feedback is the key to any success.

When problems in fluid mechanics are considered, the rationale for such procedure has been ignored due to numerous additional complications. For example, among the list that makes it difficult to extend is the fact that (a) the system is governed by partial differential equations in lieu of the ordinary variety and it has always been thought that (b) it is only the control of nonlinear effects that will have any success for flow control. And, then there is the blatant fact that the linear problem in this area has more than its share of difficulty.

The examination of perturbations to a basic flow has a long history and, for the most part, has dealt almost unequivocally with the question of the stability of the flow. This task has been accomplished but the results have not been directly useful to any sort of flow control based on first principles. Indeed, practically all of the devices that have been used are due to ad hoc considerations. For the boundary layer, unstable Tollmien-Schlichting waves are the center of attention for this purpose and hence such schemes as riblets or heating elements, for example, have been used. The question remains as to whether or not there may be other more meaningful physical means for this purpose. And, if so, how can such a means optimize the control? The answers can only be found if the full dynamics can be ascertained.

A newer analytical basis is presented that permits analysis for solution of the perturbation problem for any arbitrary initial-value specification. Then, both the transient and the long time behavior of any disturbance can be realized. With this knowledge, feedback can be designed that will suppress undesired output. Optimization of the system is then possible.

The presentation will concentrate on the method of analysis and outline the needs for determining suppression by feedback.

[1] Applied Mathematics
University of Washington, Seattle, Washington,USA

Effect of Convective Motion of Liquid Metal on Crystal Growth

A.S.Eldarkhanov[1], V.A.Efimov[1]

At the solidifying of liquid metal alloys in the mold, an intensive convective movement of melt along the zone of growing crystals is observed. The speed of these metal flows is determined by the thickness of a borderline layer distributed along the section of the mold depending on the Grashof number, which in this case equals $\delta=5.34/Gr^{0,25}$ (see Figure 1). The Figure presented shows that both the thickness δ, and the speed of sinking heated metal V from the higher horizon to the lower one is constantly growing depending on the value of h.

The investigation of temperature fields in the melt and the calculation of major parameters of the natural convection in the limited mold volume showed that the convection speed along the height of the crystallizer or the mold increases faster then the value of δ. For the crystallizer with the height of H =2000mm, V of the flow at the lower horizon increases almost two times, while δ only by 20%.

The value of the heat transfer ratio at the laminar movement of the overheated flow along the vertical edge is usually determined based on the Nusselt criterion $Nu = 0,6(Gr \cdot Pr)^{1/4}$. For all metals Pr ~ 1, while the ratio of the hydrodynamic δ to the thermal δ_t at a moderate speed of the flow movement is $\delta/\delta_t=Pr^{0.5} \sim 1$. If on the basis of the ratios presented above values of heat transfer ratios are determined at the height of $h_1 = 500$ mm and $h_2 = 2000$ mm, it proves that the value of the ratio of the to the mold being cooled at the lower horizon almost ten times exceeds its value at the upper horizon. The fast cooling and crystallization processes in these locations may be explained by an increase in V and the cooling of small volumes of liquid metal sinking due to the convection. The result obtained is of great practical importance, since it allows to ensure a rational heat transfer regime during the production of high quality castings and continuous cast ingots.

The heat transfer to the mold may also be significantly intensified by the vibration put on the hardening melt. The recurring positive and negative liquid metal pressure on the vertical crystallization zone causes the frequently observed detachment of the borderline layer and a significant increase in the speed of the flow movement at the hardening surface (see Figure 2). The assessment of the efficiency of the vibrating treatment of hardening alloys proved the possibility of obtaining even dispersed crystal structure equally along the height of the cast item.

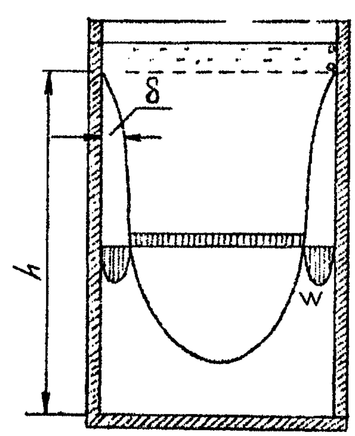


Figure 1. The distribution of the speed of convective flows in molds.

[1] *Physico-Technological Institute of Metals and Alloys National Academy of Sciences of Ukraine*

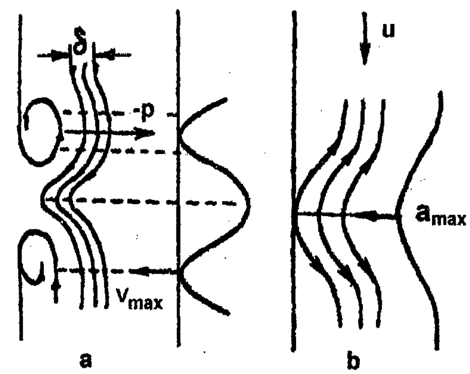


Figure 2. The detachment of the borderline layer at the moment of exhausting (a) and the growth of the pressure (b) on the borderline layer affected by the vibration wave.

Role of Turbulence Scale in Bypass Transition

E.Ya. Epik[1]

Bypass laminar-turbulent transition (BLTT), developing in boundary layer (BL) in the presence of disturbances of different nature, is broadly spread in various technical applications. As it was firstly shown in experimental investigations of IET NASU, the mechanism of BLTT at high free stream turbulence (Tu>0) has a different character from that at Tu=0: in particular Tollmien-Schlichting waves typical for LTT were nor observed meanwhile powerful fluctuations with a continuous spectrum and a maximum of kinetic energy of turbulence at $y/\delta \sim 0.3\text{-}0.4$ were present (y is distance from wall, δ is thickness of hydrodynamic BL). This type of BL, combining the features of laminar and turbulent BL (LBL and TBL), was called by us «pseudolaminar» BL (PLBL).

When Tu>0, PLBL can exhibit a number of specific pecularities, first of all, the increase of transport coefficients (friction and heat transfer) and all characteristic thicknesses of BL at the simultaneous decrease of shape parameters in profiles of velocity and temperature. The similar changes can take place in TBL, named in the same studies «quasiturbulent» BL (QTBL).

If the development of BLTT is preceded by PLBL and followed by QTBL, the broadened chain of transition, consisting of 5 links (LBL-PLBL-BLTT-QTBL-TBL) or their combination, is observed.

In investigations of IET NASU the appearance of PLBL and QTBL is connected not only with arising of initial disturbances and their development along the length of working surface but with influence of Tu and its characteristic scale, the latter being the dissipative scale L of longitudinal velocity fluctuation L. As shown by studies pointed above at certain values of relative scale L/δ .an intermediate region named «overlayer» can appear between an external flow and an outer edge of a hydrodynamic BL. The overlayer formation results in decreasing kinetic energy of normal fluctuations and weakening the transport properties at $y=\delta$ in comparison with those in an external flow.

Depending on the L/δ values there are two main cases. In the first case at high values of relative scales ($L/\delta > 1.25$) despite a high level of Tu profiles of velocity and temperature in their wall part undergo no perceptible changes, i.e. no increase in coefficients of friction and heat transfer. The BLTT starts and ends at values of transport coefficient corresponding to LBL and TBL. The false impression that PLBL and QTBL are absent can take place, however in reality it is a manifestation of the scale effect. In the second case when relative scales are comparable with the thickness of BL or less it ($L/\delta < 1.25$) the existence of PLBL before BLTT and QTBL after BLTT takes place and leads to growth of transport coefficients; BLTT can exhibit the tendency to disappearance of nonmonotonicity in variations of transport coefficients first of all heat transfer. The such case was observed in investigations of IET NASU under conditions of combined influence of Tu and closed laminar separation and was named by us «upper». It may occur in the passage part of gas turbines near the leading edge on the suction or pressure side or both. It is interesting to note that in case of upper BLTT the mechanism of influence of relative scale on transport coefficients before and after BLTT preserves, i.e. origin of overlayer is directly connected with the L value and the δ growth in the presence of separation.

It is reasonable to put a question: how does the scale influence on the location of BLTT? As shown in numerous investigations the main mechanism of LTT and BLTT is determined by amplification of longitudinal velocity fluctuations along the working surface. This fact permits to conclude that an origin of overlayer, where only changes of kinetic energy of normal fluctuations take place, can not substantially influence on the start and end of BLTT. However this problem can be solved on the basis of further experimental investigations.

[1] Institute of Engineering Thermophysics of National Academy of Sciences of Ukraine (IET NASU), 2a Zhelyabov Street, Kyiv, 03057, Ukraine; e-mail: epik@epik.kiev.ua

Canopy Vorticity Seen from Spectrum Measurements

Ye.A.Gayev[1], E.Savory[2], N.Toy[3]

The terms "***canopy***" or "***penetrable roughness***" (PR) flow express common features of air or water flows interacting with forest and agricultural plants (Meroney, Dubov ea, Raupach ea), vegetated areas in river beds (Kouwen, Nuding, Bennovitsky) or large-scale atmospheric spraying systems (Porter & Chen, Gayev ea [3]). Until recently, our knowledge about these canopy (or PR) flows has been quite sufficient, such that the mathematical models developed so far may generally be employed in practical problems when one is only dealing with mean flow quantities. However, for explaining some unexpected turbulence features found in natural forests (Shaw, Finnigan [1]) and for solving canopy diffusion problems, the nature of the vorticity originating within the canopy and spreading outwards into the external flow must be taken into consideration. The present work continues our previous experimental canopy research [4] and concentrates on the vortical flow features represented through spectrum measurements of longitudinal flow fluctuations. It is hardly possible to reveal vortices and coherent structures in such highly vortical and turbulent air flow by means of visualisation. Spectral analysis of longitudinal velocity pulsations was used in our research to compare vorticity field within and over tall penetrable roughness with those over a relatively smooth surface and the flow past an isolated PR elements.

Experiments were carried out in a wind tunnel at the University of Surrey, UK which has working section dimensions of 1.37m width x 1.68m height [4]. No unexpected phenomena were found over smooth surface (wind tunnel bottom), and this validates thermal anemometry technique and acquisition algorithms employed. For wind tunnel speed 8 m/s, spectral curves were quite close and similar to one another for elevations $0.03 \le z/\delta \le 0.7$ where boundary layer thickness was $\delta = 75mm$. As usual, they consist of energy containing $1 < f < 200Hz$, inertial subrange $20 < f < 500Hz$ with $E \sim f^{-5/3}$, and dissipation subrange $700 < f < 3000Hz$ with $E \sim f^{-4}$. Energy of vortices sharply decreases far away from the surface and became almost comparable with hot wire electrical noise for elevation $z = 1.3\delta$.

A number of obstructions in a form of "trees" was constructed with a cylindrical "stem" (h_1 is its height) and a slotted flat-plate "crown" in the shape of an equilateral triangle (h_2). Three-dimensional wake behind such a single obstruction has evidently inhomogeneous structure with respect to appearance of the spectrum curves because the "crown" facing the flow was of significant "equivalent diameter" approximately $5.6cm \approx 1.3\delta$. Fluctuation history was collected in many elevations z on verticals projected to various x- and y-locations of the underlying wind tunnel bottom. In many space points behind "crown" edges, remarkable peaks were found on spectrum curves. They were evidently caused by vortices shedding from edges like it was shown in [2].

The canopy with a total height $h=h_1+h_2=120mm$ was formed from a staggered array of such "trees". The streamwise and lateral spacings, Δx and Δy, between them was varied in the models investigated thus providing penetrable roughness setups of different densities. Vortices in the flow are born behind each individual obstacle constituting the canopy when the flows runs around them. Infinite number of separate vortices interact with each other within the canopy. Such extremely intensive mixing of flowing medium leads to relatively unique distributions of mean flow quantities, especially far enough from the canopy inlet as shown in [4]. It is just the main question how the plenty of vortices within the obstacle layer interact with each other, how they produce resulting turbulence and spread outwards the PR into the external flow. That is why spectrum behaviour was investigated throughout obstacle layer, within and over it, for several PR-fetch lengths. Figure 1 provides several spectrum curves to highlight their most important features. Spectrum curves for elevations z=2, 20, 40, 60, 100 and 120mm=h may be seen to almost coincide. This indicates rather homogeneous vorticity within the obstacle layer depth despite inhomogeneous structure of the canopy itself consisted from rare "trunk

[1] *Institute of Hydromechanics UNAS, Zhelyabov str., 8/4. 03057, Kiev, Ukraine.*

E-mail: gayev@gayev.pp.kiev.ua

[2,3] *Fluid Mechanics Research Group, Dept of Civil Engineering, University of Surrey, Guildford,, GU2 7XH, UK.*
E-mails: e.savory@surrey.ac.uk , n.toy@surrey.ac.uk

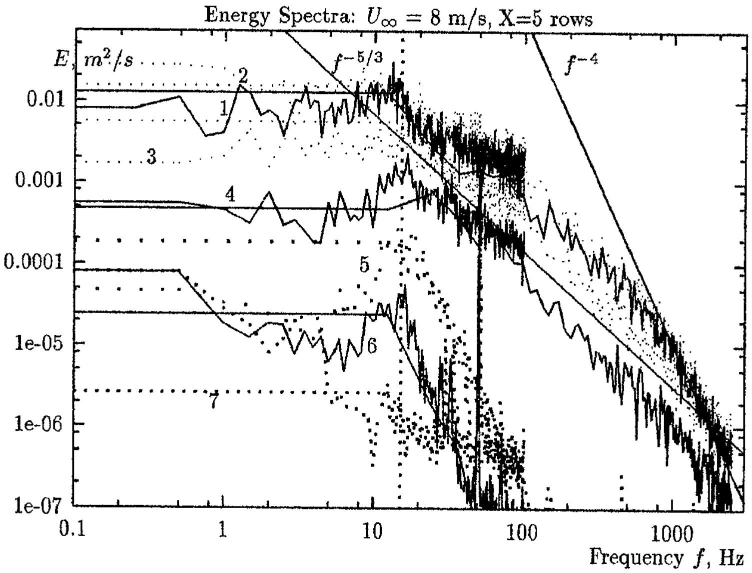


Figure1:

Change in spectrum curves within the PR obstacle layer and over it on elevations indicating existence of coherent structures: 1 - z=20mm; 2 - 120=h; 3 - 220; 4 - 240; 5 - 300; 6 - 360; 7 - 605mm

space" and dense "crown space". Energy distribution by frequency has been somewhat distinguishing from mentioned over (it is constant for "slow" vortices 0<f< 10 – 15 Hz, decreases as $f^{-5/3}$ in inertial region and as f^{-4} in dissipation region 900<f<3 000 Hz) but no specifically pronounced features were found. Note that frequency peaks behind individual obstructions have been disappeared.

Spectrum curves began slowly go down with further elevation over surface with the PR; no significantly new features were observed up to z=220mm=1,8h. However, drastic changes in the longitudinal fluctuation spectrum shape happened with further distance over the PR: a small peak at f~15Hz appeared on the curve and continued to grow with further elevations up to z~420mm=3,5h. Then peaks reduced, and wall over the penetrable roughness they disappeared totally and spectrum curve calmed down thus indicating the undisturbed external flow.

Such behaviour of spectrum curves over the canopy (or the PR) evidently indicates a principal difference of canopy vorticity and turbulence from that over "empty" surface. Finnigan ea [1] explored resembling with mixing-layer case flow to "extract" an image of coherent vortical structures, with sweeps and ejections, being formed over the canopy. Our findings really confirm the principal difference between internal and external vorticity they pointed out.

High turbulence level within the PR (up to 60%), decelerated flow within the obstacle layer with an inflection point on mean velocity profile, possible stagnation zones etc provide specific flow features that may be implemented in technical devices for controlling boundary-layer flow.

References

1. **Finnigan J.** Turbulence in plant canopies. *Ann. Rev. FluidMech., 2000, v. 32, pp. 519 – 571.*
2. **Fox T.A.** PhD Thesis. *Surrey university, (UK)*, 1986.
3. **Gayev Ye.** Aerothermal theory of easily penetrable roughness: Particular application to the atmospheric flow in and over longscale spray cooling system, *Il Nuovo Cimento, 1997.* 20C, *No.* 3: 331-342.
4. **Gayev Ye. A., Savory E., Toy N.** Investigation of a inhomogeneous penetrable roughness in wind tunnel. *Applied Hydromechanics* (Kiev), *No.*1, *2(74), 2000 (in Russian).*

Theoretical Study of Control Techniques That Use Generation of Cross Vortices in a Near-Wall Flow

V.Gorban, I.Gorban[1]

It is known that near-wall flows are governed by evolution of vortices. Therefore controlling the vortical pattern of a turbulent boundary layer one acts on the main characteristics of the flow under consideration. The strategy based on artificial generation of large-scale stable vortices is one of the most perspective approaches used for a control of near-wall flows. In this case, one deals with a creation of «intellectual» flow of fluid, in which the vortices are formed according to both the control scheme and either theoretical or semiempirical model that predicts the vortex behavior. If the vortices created are stable, the level of chaotic motions in the flow drops that can lead to decreasing the friction near a body, delay of flow separation, rising of the lift force of a wing.

In the present work, the numerical analysis of some techniques that can be applied to generate cross stable vortices in a near-wall flow is performed. It is based on the model of ideal incompressible fluid and uses the discrete vortex representation of the flow field for a two-dimensional problem. The areas of continuous vorticity generated in the flow are replaced by assemblies of the inviscid discrete vortices which move as fluid particles. It is generally accepted that the free vorticity is formed near sharp edges of the flow boundary only and effect of viscosity on generation and evolution of large-scale motions is secondary. We consider two patterns of the flow with artificial vortices.

In the first case, the periodical system of the vortices that move in a near-wall flow is generated. This technique can be used for improvement of hydrodynamic loads of a wing. The flow scheme is depicted in Fig. 1 *a*. The generator of vortices was supposed to be mounted near the leading edge of wing. In the calculations, the wing was simulated by the thin plate. Analysis of the obtained results demonstrates that the present technique ensures large enough increment of the lifting force of wing (Figs. 2, 3). The efficiency of the scheme being to depend on the following dimensionless parameters: $\Gamma^* = \Gamma / U_\infty b$, $T^* = TU_\infty / b$, where U_∞, b, T are the free-stream velocity, the chord of the wing and the period of vortex generation respectively. Fig. 2 shows the coefficient of the wing lifting force C_y against the angle of attack α at different values of the circulation of the artificial vortices (here curve **1** corresponds to $\Gamma^* = 0$, **2** – $\Gamma^* = -.25$, **3** – $\Gamma^* = -.5$, $T^* = 1$). The functions $C_y(T^*)$ represented in Fig. 3 at the different angles of attack indicate an occurrence of optimal frequencies for generation of the large-scale vortices near the wing (here $\Gamma^* = -.5$, **1** – $\alpha = 5^\circ$, **2** – $\alpha = 10^\circ$, 3 – $\alpha = 15^\circ$).

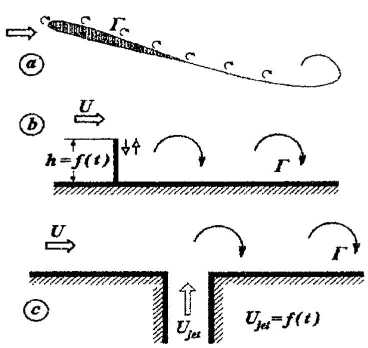


Fig. 1.

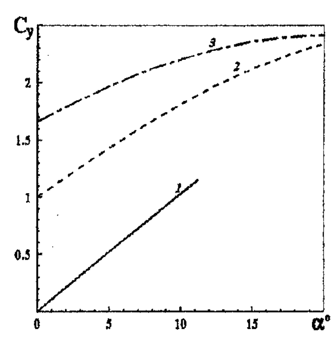


Fig. 2.

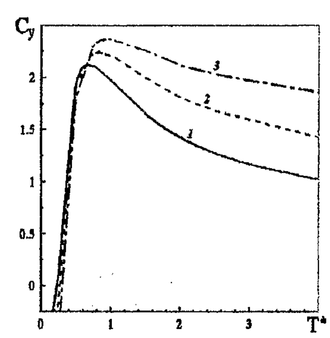


Fig. 3.

Efficiency of the control technique proposed above is defined by stability and survivability of the generated vortices and depends on work of the vortex generator used in the system. Here we present

[1] Institute of Hydromechanics, Kiev, Ukraine
Address: 8/4, Zhelyabov Str.
E-mail: ira@gorban.pp.kiev.ua

results of the numerical simulations dealing with activity of vortex generators of two types. Those are the interceptor which moves periodically up and down and the jet with a periodic change of its rate. The schemes of the vortex generators are depicted in Figs. 1 *b* and 1 *c*. Here $f(t)$ is the periodic function. The present calculations provide estimations for optimal lows describing the interceptor motion and change of the jet rate. They point out that interceptor oscillations of high frequency lead to intensive interaction of the vortices that have been generated before one with other. This process causes breaking of the regular vortex system and, as a result, fast chaotization of the flow. The flow pattern corresponding to this case is shown in Fig. 4. Here the dimensionless period of vortex generation $TU_\infty/h_{\max}=3$, where $h_{\max}$ is the maximum height of the interceptor during the period. The calculations shown the stable vortex system is formed under condition $TU_\infty/h_{\max}>6$. Fig. 5 corresponds to the case when $TU_\infty/h_{\max}=12$. Note that Figs. 4, 5 include the laws of the interceptor motion $h(t)$. Analogous restrictions imposed to the frequency of the vortex generation are also valid for the oscillating non-stationary jet that is injected into a near-wall flow (Fig. 6).

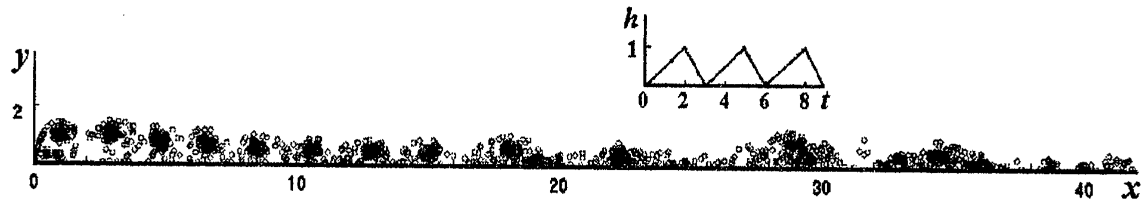


Fig. 4.

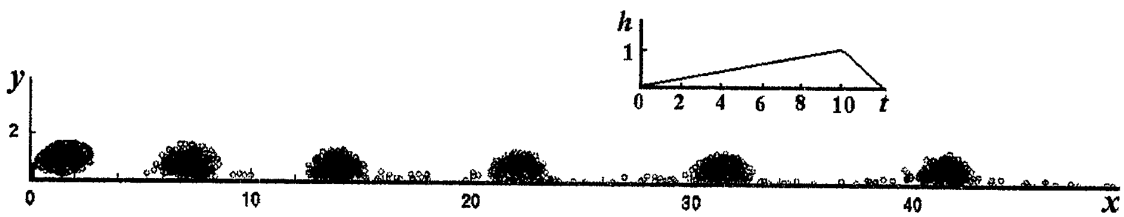


Fig .5.

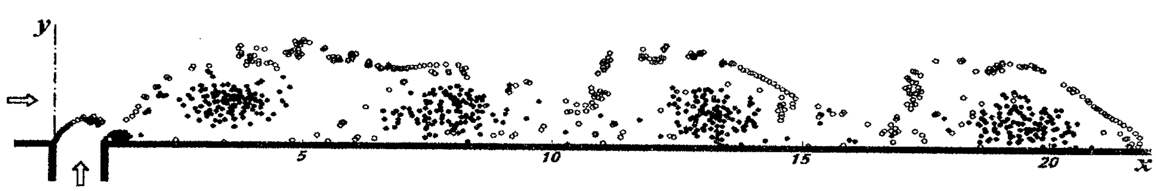


Fig. 6.

Another class of the control methods under consideration deals with generation of a system of standing vortices, or stationary circulation zones, near a body. Their efficiency depends on stability of the vortices. The numerical analysis carried out in the present work demonstrated that stabilization of the standing vortices generated by interceptors and cross grooves may be obtained with applying fluid bleed systems. The calculations denote an importance of the configuration such vortex generators as well as of location of the sink in the system and its intensity.

Transitions in the Viscous Vortex Ring

F. Kaplanski[1] and Ü. Rudi[2]

Keywords - Vorticity - Vortex Ring – Transitions in Flows

Abstract – The elementary vortex structure, like a vortex ring, is the fundamental object in the theory of vortex flow. The compact nature of this structure also makes it ideal as simpler building block in the modeling of more complex flows, including axisymmetric turbulent jets (Ashurst, 1983) and boundary layers (Adrian, 1990).

The analysis shows that a uniformly valid second approximation of the solution for the problem of the viscous vortex ring can be obtained after impressing a spatially uniform drift on the first-order solution (Rott and Cantwell, 1993). The new model of the vortex ring is developed by making use of this result and through the applying of the existing solution of the Stokes equation. The flow is assumed to be axisymmetric and incompressible with constant density ρ and viscosity ν. A earlier found solution for the limit $Re \to 0$ is given by the expression for the vorticity

$$\omega = \exp\left(-\frac{1}{2}\left(\sigma^2+\eta^2+\tau^2\right)\right)I_1(\sigma\tau),$$

where the dimensionless variables are

$$\sigma=\frac{r}{L},\ \eta=\frac{x-x_o(t)}{L},\ \tau=\frac{R_o}{L},\ \omega=\frac{\varsigma}{\varsigma_0};\ L=(2\nu t)^{1/2},\ Re=\frac{\varsigma_0 L^2}{\nu}$$

and I_1 denotes the first-order modified Bessel function of the first kind. The parameter $x_0(t)$ is the distance, which the vortex ring passes from the initial moment t_0 and R_o designates the ring radius at t_0. The flow invariant is the impulse of vorticity

$$I=\pi\rho\int_0^{\infty}\int_{-\infty}^{\infty} r^2\varsigma\, dx\, dr$$

with the help of which we can obtain

$$\varsigma_0=\frac{2M}{(4\pi\nu t)^{3/2}R_0},\ M=\frac{I}{\rho},$$

$$Re=\frac{M}{2(\pi\nu)^{3/2}(t)^{1/2}R_0}=\frac{M\tau}{2^{1/2}(\pi)^{3/2}\nu R_0^2}=\frac{Re_0\,\tau}{\sqrt{2\pi}},$$

where $Re_0=\frac{\Gamma_0}{\nu}$ is the initial Reynolds number and $\Gamma_0=\frac{M}{\pi R_0^2}$ is the initial circulation.

In the long-time limit the considered distribution transforms into the self-similar Phillips' distribution and for $t \to 0$ it tends to delta-function. The method of integral transforms allows us to derive the corresponding velocity field inside the moving vortex ring:

$$u_t=\frac{1}{\sigma}\frac{\partial\Phi}{\partial\sigma}=\frac{\sqrt{\pi}}{2\sqrt{2}}\int_0^{\infty}\mu F(\mu,\eta)J_1(\tau\mu)J_0(\sigma\mu)d\mu,$$

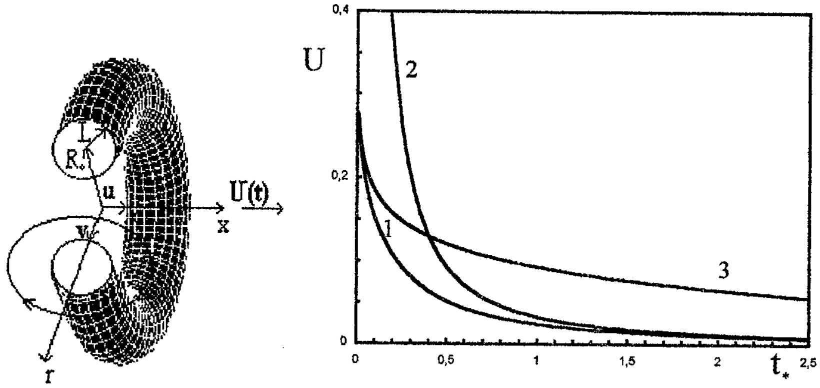
$$v_t=-\frac{1}{\sigma}\frac{\partial\Phi}{\partial\eta}=-\frac{\sqrt{\pi}}{2\sqrt{2}}\int_0^{\infty}\mu\{-G(\mu,-\eta)+G(\mu,\eta)\}J_1(\tau\mu)J_1(\sigma\mu)d\mu,$$

where $F(\mu,\eta)=G(\mu,\eta)+G(\mu,-\eta)$, $G(\mu,\eta)=\exp(\eta\mu)(1-\mathrm{erf}(\frac{\mu+\eta}{\sqrt{2}}))$ and erf (z) is the error function.

The obtained streamfunction behaves similarly to the vorticity distribution: at large time it transforms into Phillips' result and for $t \to 0$ tends to a circular line vortex. A new expression is found for the

[1,2] Department of Aeromechanics, Estonian Energy Research Institute
Paldiski mnt.1, Tallinn, 10137, Estonia
Fax: (3722) 6613655 Phone: (3722) 6621822 Email: fkaplan@online.ee

velocity of the ring, which agrees both with the long-time asymptotic drift velocity (Rott and Cantwell, 1993) and the translation velocity for rings with small cross-sections (Saffman, 1970).



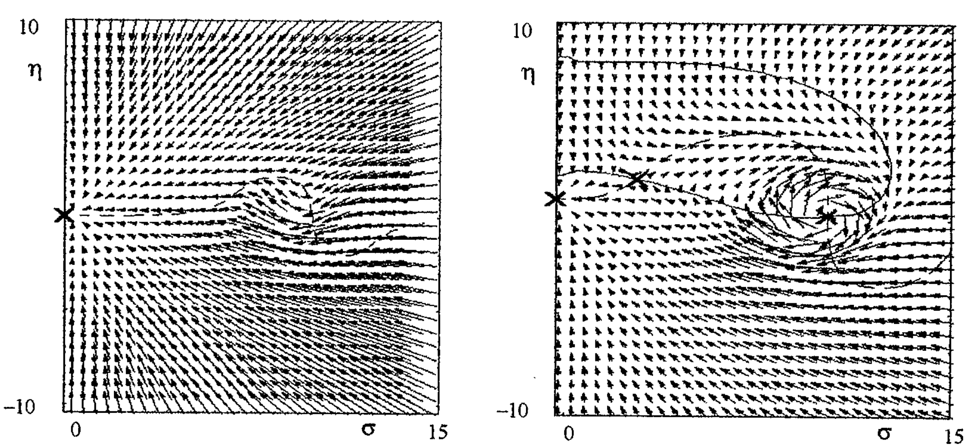
Schematic representation of the vortex ring (left) and temporal evolution of ring's translation velocity (right). Curve 1 is the obtained velocity via $t_* = 2t\nu / R_0^2$. Curve 2 shows the asymptotic drift velocity and curve 3 represents Saffman's result.

A good agreement between the derived translation velocity of a viscous vortex ring and available experimental data (Eisenga et al., 1998) confirms the efficiency of the given model. This model combines earlier found estimates for a vortex ring for small and large time and describes evolution of thin ring to thick due to viscosity.

On the basis of the obtained velocity field the system for pathlines of fluid particles is found to which the method of entrainment diagrams is applied (Cantwell, 1981). The equations for particle paths are

$$\frac{d\sigma}{ds} = -\frac{\sigma}{2} + \frac{2\sqrt{2}}{\sqrt{\pi}}\frac{\mathrm{Re}_0\,\tau}{8}v_t, \quad \frac{d\eta}{ds} = -\frac{\eta}{2} + \frac{2\sqrt{2}}{\sqrt{\pi}}\frac{\mathrm{Re}_0\,\tau}{8}u_t, \text{ where } s = \ln t.$$

In spite of the trivial Reynolds-number dependence of the obtained solution, which is valid for small Reynolds number, the resulting system exhibits a Reynolds-number dependence that is quite complex. Transition in the viscous vortex ring is examined as a bifurcation of this system. It is shown that for high value of τ for large initial Reynolds numbers three regimes of particle motion exist, and for small τ there are only two regimes. In the additional regime the particles trajectories are divided into two parts: some of them move towards a critical point lying in the axis of symmetry while other part begins to be involved in the core of a ring due to the growth of concentration of vorticity. This separation manifests the influence of nonlinearity and can result to a shedding of impulse into the wake, which is typical for the turbulent motion. The modification of the offered model for a turbulent vortex ring is considered and its application to fuel injection process is also discussed.



Entrainment diagrams. The pattern of critical points for $\tau = 10$: $Re_0 = 100$ (left); $Re_0 = 600$ (right).

Controlling a Linear Process in Turbulent Channel Flow

John Kim[1] and Junwoo Lim

Near-wall streamwise vortices have been recognized as the most relevant turbulence structure to control from a perspective of drag reduction in turbulent boundary layers (Kim, 1992). In the present work we consider a new approach of controlling these streamwise vortices for the purpose of drag reduction in turbulent boundary layers and transition delay in laminar boundary layers. This new approach is based on the observation that a linear process is responsible for the formation of these streamwise vortices (Butler and Farrel, 1993; Kim and Lim, 2000).

The transient growth due to non-normality of the linearized Navier-Stokes system has received much attention during the past several years. It has been shown that certain disturbances can grow to $O(Re^2)$ in time proportional to $O(Re)$. It has been postulated that the transient growth, which is a linear process, can lead to transition to turbulence at a Reynolds number smaller than the critical Reynolds number, below which a classical linear stability theory based on the modal analysis predicts that all small disturbances decay asymptotically. As such, some investigators attributed this linear process as a possible cause for subcritical transition in some wall-bounded shear flows.

Some investigators further postulated that the same linear process is also responsible for the observed wall-layer streaky structures in turbulent boundary layers. The notion that commonly observed wall-layer structures are related to a linear process suggests that the same linear process may play an important role in fully nonlinear turbulent boundary layers.

The role of this linear process in fully nonlinear turbulent flows is examined to explore whether controlling the linear process is a viable approach for turbulence control. It is found that the linear process associated with the coupling term plays an important role even in fully nonlinear wall-bounded turbulent shear flows. Near-wall streamwise vortices, which have been found to be responsible for high skin-friction drag in turbulent boundary layers, could not be sustained without the coupling term. The fact that the coupling term plays an essential role in maintaining these streamwise vortices suggests that an effective control algorithm for drag reduction could be aimed at reducing the effect of the coupling term in the wall region. In fact, the opposition control used by Choi *et al.* (1994) could be viewed as a control scheme trying to reduce the effect of the coupling term by suppressing the spanwise variation of the wall-normal velocity in the wall region.

The present result is consistent with recent studies, in which several researchers (Joshi *et al.*, 1997; Bewley and Liu, 1998; Cortelezzi *et al*, 1998) have shown that controllers designed from a linear system theory work surprisingly well in reducing the viscous drag in turbulent boundary layers. These results suggest that the essential dynamics of near-wall turbulence could be approximated through the linearized model. Also discussed will be how to design a reduced-order controller that directly accounts for the coupling term in the cost function to be minimized. Other control approaches used by other investigators are also examined from the point of the linear systems theory.

References:

Bewley, T. & Liu, S. (1998), "Optimal and robust control and estimation of linear paths to transition," *J. Fluid Mech., 365.*

Butler, K. & Farrell, B. F. (1992), "Three-dimensional optimal perturbations in viscous shear flow," *Phys. Fluids A, 4, No. 8.*

Choi, H., Moin, P. & Kim, J. (1994), "Active turbulence control for drag reduction in wall-bounded flows," *J. Fluid Mech., 262.*

Cortelezzi, L., Lee, K-H., Kim, J. & Speyer, J. (1998), "Skin-friction drag reduction via robust reduced-order linear feedback control," *Int. J. Comp. Fluid Dynamics, 11.*

Joshi, S., Speyer, J. & Kim, J. (1997), "A systems theory approach to the feedback stabilization of infinitesimal and finite-amplitude disturbances in plane Poiseuille flow," *J. Fluid Mech., **332**.*

Kim, J. (1992), *"Study of turbulence structure through numerical simulations: the perspective of drag reduction,"* in AGARD Report (R-786), AGARD FDP/VKI Special Course on "Skin Friction Drag Reduction," March 2-6, 1992, VKI, Brussels, Belgium.

Kim, J. & Lim, J. (2000), "A linear process in wall-bounded turbulent shear flows," *Phys. Fluids, **12**, No.8.*

[1] *Department of Mechanical and Aerospace Engineering, University of California, Los Angeles Los Angeles, CA 90095-1597*
310-825-4393 (voice), 310-206-4830 (fax), jkim@seas.ucla.edu

The Complex Influence of a Surface Pliability and of High Molecular Weight Polymer Additives on Turbulent Friction

V.I.Korobov[1]

As a result of evolution in nature of flying and swimming objects, various mechanisms and adaptations have been developed to increase performance or speed that helps a survival (life-support) of species in an ecological niche. The singularities in exterior shape geometry of hydrobionts promote a diminution of separation zones and drag reduction of the form. Morphological features of skin structure provide turbulent friction reduction. They are rather variable and have an effect on different areas of a boundary layer (BL) [1,2]. In animate nature, several mechanisms are simultaneously realized and work combined for decrease power losses at motion in natural environment.

Various methods of the near-wall (NW) flow control are also known in engineering [3,4,5,6]. One of them utilizes various effects due to compliance of the streamlined surface (visco-elastic properties) [6,7,8,9]. Other important class of NW turbulent flows control is determined by adding into fluid an insignificant concentration of high molecular weight polymer (HMP) components – Toom`s effect [10]. However, combination methods of BL control have not been practically investigated yet. Attempts to explain complex mechanism of mutual action of these factors onto BL were made in works [9,11,12,13,14].

The purpose of the present researches is to evaluate experimentally the influence of modified boundary conditions, namely, joint influence of the wall pliability and small additions of a HMP water solution in the NW area on integral performance of BL in comparison with those in the flow over a rigid surface.

The measurements of hydrodynamic friction of longitudinally stream-lined the rigid cylinder and cylinder with the modified boundary conditions were made with tensodynamometer at towage in a hydrochannel, in the range of velocities $U = 2.0 \div 22.0$ m/s. Diameter of the cylinder is $d = 0.175$ m, lengthening of the cylinder with a nose fairing is $\lambda = L/d = 6.07$.

Metal cylinder with polished outside surface is used as a rigid wall. As a pliable wall, the cylinder with elastic coating from polyurethane of thickness $t/(d/2) = 1.71 \cdot 10^{-2}$ and density $\rho_e = 1250$ kg/m^3 is used. The measurements data for dynamic visco-elastic performances of elastomer in the frequency band $0 < \omega < 320$ s^{-1} are given in the paper [8]. The static and dynamic modules of an elastomer elasticity are accordingly equal to $1{,}6 \cdot 10^3$ and $5 \cdot 10^3$ KPa. Factor of mechanical losses at frequencies up to 100 s^{-1} was about 0.53, and at frequencies up to 300 s^{-1} increased up to 0,7.

Water solution of polyoxiethylene (POE) with a molecular weight $M_\omega = 4 \cdot 10^6$ at weight concentration $\alpha = 10^{-3}$ was applied as HMP components. The polymer solution was injected in BL tangentially to the stream-lined surface through the ring slot of $s = 3.0 \cdot 10^{-4}$ m thick in the nose fairing. Two series of experiments (*A* and *B)* are conducted with rigid and compliant surfaces, which differ by total amount of non-Newton liquid injected. The magnitude of the POE volumetric consumption coefficient $C_q = Q/US$ was varied in dependence on Reynolds number. For series A : $C_q \sim 5 \cdot 10^{-5}$ at Re $\sim 2 \cdot 10^6$, $C_q \sim 3 \cdot 10^{-5}$ at Re $\sim 10^7$ and $C_q \sim 2 \cdot 10^{-5}$ at Re $\sim 2 \cdot 10^7$. For series B : $C_q \sim 4 \cdot 10^{-5}$ at Re $\sim 2 \cdot 10^6$, $C_q \sim 2 \cdot 10^{-5}$ at Re $\sim 10^7$ and $C_q \sim 1.5 \cdot 10^{-5}$ at Re $\sim 2 \cdot 10^7$.

Measurements data shown in the figure 1 are integrated effect magnitude ξ(Re) as functions of from the Reynolds number. $\xi(\mathrm{Re}) = (C_{x\,st} - C_{x\,i})/C_{x\,st}$, where $C_{x\,st}$ is the friction coefficient of a standard rigid surface; $C_{x\,i}$ is the friction coefficient of cylinder with the modified boundary conditions.

The effect is higher in the case with higher HMP additive supply. So if $C_{qA} > C_{qB}$ is set, then accordingly is registered: for the standard – $\xi_2 > \xi_1$; and for compliant surface – $\xi_4 > \xi_3$. Here the index at x corresponds to curve number in the figure. At combined action, the magnitude of hydrodynamic effect in comparison with applying of single factor is obtained: $\xi_3 > \xi_1$ и $\xi_4 > \xi_2$. At that, the an effects additivity it is visible [12]. Thus, with allowance for to an experiment error, $AC + AB = AE;\ AC + AD = AG.$

[1] Institute of Hydromechanics NASU, Zhelyabov str., 8/4. 03057, Kiev, Ukraine

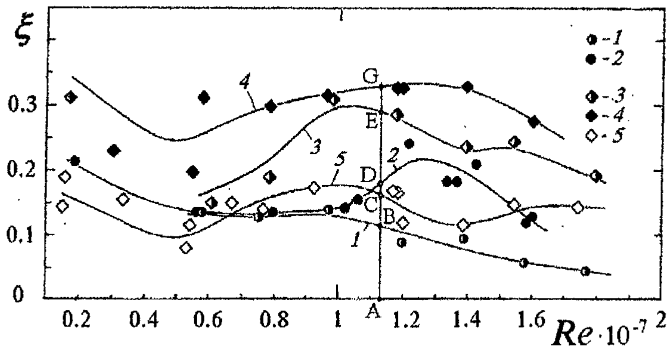


Figure 1. The drag reduction coefficient ξ(Re) on Reynold's numbers: at injection of the polymeric components in a BL on rigid (*1*, *2*) and elastic (*3*, *4*) surfaces; 5 - without the polymeric components (C_q= 0) on an elastic surface. For experimental curves *1*, *3* laws of POE feeding corresponded to the flow rate C_q (Re) under the law "B"; for a curve *2*,*4* - under the law "*A*".

The introduction of polymeric molecules associatives makes main contribution to modification of Reynolds stresses near the wall (in buffer zone). One from developments of visco-elastic coating action on a TBL is evaluated as a corollary it of work in quality of an energy-absorbed wall, responding on action of large-scale structure of a NW turbulence. Besides it is rise stability of longitudinal vortical structures. The toting of effect can serve indirect confirmation of mechanisms of action of the indicated factors on a BL. The complex action has complicated and nonlinear character. Thus some modification of a microstructure statistically of ordered flows in NW area is possible also.

References

1. **Bushnell D.M., Moore K.J.** Dgag reduction in nature // *Annu. Rev. Fluid. Mech.- 1991.-23.- P 65-79.*
2. **Pershin S.V., Chernishov O.B., Kozlov L.Ph., Koval A.P., Zaets V.A.** Regularity in covers of high-speed fishes // *Kiev: Bionics- 1976.- 10.- C. 3-21.*
3. **Basin A.M., Korotkin A.I., Kozlov L.Ph.** Control by a ship boundary layer .- Leningrad: Shipbuilding, 1968.- 391 P.
4. **Chang P.K.** Control by a stream separation.- Moscow: Mir, 1972.- 552 P.
5. Some problems by near-wall flows control / **V.G. Bogdevich, G,Ph. Kobets, G.S. Kozuk, G.S. Migirenko, V.I. Mikuta, B.P. Mironov, B.G. Novikov, V.A. Tetianko, Yu.V. Shtatnov** // *Appl. Mech. and Techn. Phys.- 1980.- № 5, - C. 99-109.*
6. **Bushnell D. M.** Turbulent drag reduction for external flows // *AIAA Pap.-1983.- N 83-0227.- P. 1-20.*
7. **Kramer M.O.** Boundary layer stabilization by distributed damping // *J. Am. Soc. Naval Eng.-1960.- 72, № 1.- P. 25-33.*
8. **Korobov V.I., Babenko V.V.** About one mechanism of interaction of an elastic wall with a stream // *Eng.-Phys. J.- 1983.- 44, № 5.- C. 730-733.*
9. **Babenko V.V., Canarsky M.V., Korobov V.I.** The boundary layer on elastic plates.- Kiev: Naukova dumka, 1993.- 264 P.
10. **Pilipenko V.N.** Influence of the additives on near-wall turbulent flows.- The totals of a science and engineering: Mech.Ffluid and Gas.- *Moscow: VINITI of AS USSR, 1980.- 15.- C. 156-257.*
11. *Patent of USSR № 483538, Kl. F 15D 1/12.* Damping coating. **Babenko V.V., Kozlov L.Ph., Korobov V.I.**.- *Publ. 05.09.75, Bul. № 33.*
12. **Korobov V.I.** About joint influence of a compliant surface and polymeric additives on integrated performances of a boundary layer // *Proc. III conf. "Problems of a hydromechanics in opening up of ocean".- Kiev: IHM NASU, - 1984.- C. 243.*
13. Influence of a flow velocity at a joint operation of a compliant surface and polymeric components on turbulent friction / **V.M. Kulic, I.S. Poguda, B.N. Semenov, T.I. Yushmanova** // *Izvestija Sib. Branch AS USSR- 1987.- 4, № 15.- C. 42-46.*
14. **Korobov V.I.** Combined influence of a surface compliantivity and polymeric additions on a boundary layer // *Kiev: Bionics.- 1993.- 26.- C. 27-31.*

Numerical Simulation of the Dense Gas Plume Effects on the Atmospheric Boundary Layer Structure

I. Kovalets[1], V. Maderich

An increased public concern has expressed over past few decades for the hazards associated with the manufacture, storage, transport and use of large amount of chemicals in gaseous or liquefied form. In many accidents the releases produced dense gas plumes. Their dispersion is strongly affected by buoyancy forces and coupled with the dynamics of the near-surface atmospheric turbulent layer. To help understand and simulate the dispersion processes the comprehensive three-dimensional models are necessary. These models should describe main physical factors that govern dispersion of dense gas plumes.

In this paper we present the numerical model of dense gas cloud dispersion (Kovalets and Maderich, 2000). The system of equations of mass, momentum, energy and state coupled with turbulence sub-model is used. The distinctive properties of model are:

(i) Use of Favre-Reynolds averaging procedure that results in the less complicated form than the Reynolds procedure;

(ii) The energy equation was rewritten in terms of pressure that has advantage for numerical solution of problem;

(iii) The modified for stratified compressible flow standard $k-\varepsilon$ model with algebraic relations for turbulent stresses and fluxes;

(iv) Two ordinary equations for surface temperature and thickness of ground layer are included to parametrize the heat transfer in the ground.

The eight differential equations in particular derivatives for three components of mean velocity, density, pressure, concentration, turbulent kinetic energy and dissipation, together with appropriate initial and boundary conditions, are discretized by the finite-difference method in space and time. They are fully implicit in time. The splitting method upon the space directions and physical processes is used. The two-stage procedure is used for $n+1$ step. At first stage the velocity and pressure are calculated. At second stage density, concentration and turbulence characteristics are calculated using values of velocity at $n+1$ step. This approach allows using of larger time steps at second stage and improves mass conservation comparatively with use of enthalpy equation (Kovalets and Maderich, 1999).

The results of calculations are compared with the results of laboratory field experiments It was shown that the "anelastic approximation" failed in initial stage of dense gas plume collapse. In the calm atmosphere the plume formed vortex ring that spread in self-similar manner. The analysis of the energetics of the dense gas dispersion processes showed that at first stage the potential energy passed to mean kinetic and in the turbulence. The turbulent energy decays faster than potential energy and mean kinetic energy at the cost of dissipation and mixing that again increase the potential energy. In accordance with experiments of Zhu et al. (1998) model predicts strong inhibition of turbulence, modification of mean velocity and concentration profiles inside of plume. The vertical turbulence intensity is reduced by more than 30% and local Richardson number attained the maximum value in the same place where turbulence intensities are most reduced.

References

Kovalets I., Maderich V. (1999) Dynamics and energetics of dense gas dispersion in near-surface atmospheric layer. Applied Hydromechanics, **74**,

Kovalets I., Maderich V. (2000) Numerical modelling of heavy gas dispersion in the near-surface layer of atmosphere. Applied Hydromechanics, (submitted).

Zhu G., Arya S.P., Snyder W. (1998) An experimental study of the flow structure within a dense gas plume, J. Hazardous Materials, **62**, 161-186.

[1] Institute of Mathematical Machine and System Problems NASU, Kiev, Ukraine
e-mail: ik@env.immsp.kiev.ua

On the Spreading Mechanism of 3D Turbulent Surface Jets

Brian E. Launder[1]

ABSTRACT

The presentation examines the mechanisms proposed for the very asymmetric spreading rates found in the 3-dimensional turbulent wall jet: experiments show that the spreading rate parallel to the wall is some 5-8 times as great as that normal to the wall. As Newman et al (1972) had originally proposed the cause is the creation of strong streamwise vorticity which sweeps fluid down towards the wall and ejects it laterally parallel to thesurface. The source of this streamwise vorticity turns out to be the anisotropy of the Reynolds stresses in the plane perpendicular to the jet's axis. Thus the mechanism is the same as that which drives turbulent secondary flows in duct of non-axisymmetric cross section. However, the magnitude of the induced secondary flow is fully one order of magnitude larger than in duct flows. It is demonstrated that the spreading behaviour can be faithfully captured by a second-moment closure developed at UMIST over the past decade which, importantly, satisfies the two-component limit to which turbulence reduces at a wall. A computational study of the free surface jet suggests that that flow should have an even greater asymmetry of spreading than the wall jet though none of the available experimental data have extended far enough downstream to approach the fully-developed limit.

[1] UMIST, Manchester, UK

Vortex Generators in the System of Local Aerodynamics Improvement

Anatoliy P. Leonenko.[1]

The typical aerodynamic feature of modern high-speed airplanes is boundary layer flowdown towards the wingtips with angle-of-attack and/or Mach number increase. Boundary layer separation which occur in this case, can highly unfavorably influence stability and control characteristics and lead in particular to: loss of static stability; buffeting; aileron effectiveness decrease.

One of the radical means for suppression the unorganized boundary layer separation is a system of vortex generators on the upper surface of a wing. Their typical height is of the order of 1% of local chord length.

Vortices initiated by vortex generators and spreading parallel to upper wing surface (by the outer edge of boundary layer) promote the extensive interfusion between boundary layer and outer flow. As a result the separation region is substantially minimized.

At the same time due to ambient flow improvement the Mach-number-stability of wing profiles rises up, the aileron effectiveness grows, and buffeting eliminates.

But presence of vortex generators in many cases leads to an increase of an aerodynamic drag of the airplane. That is why in Antonov DB we prefer to perform the conditioning of local aerodynamics of the wing - the improvement of flow structure using less energy-spending methods as far as possible. For instance in the course of investigation of vortex generators aimed to increase longitudinal static stability of Antonov 124 "Ruslan" airplane at high angles-of-attack and high speed (Mach number) their retraction was provided on to the theoretical contour of the wing.

Flow separation in the aileron part of wing of Antonov 32 airplane in cruise configuration was eliminated with specially developed tetrahedron-shaped vortex generators, located along the line of 42% of chord in the aileron region of the wing.

The regions of separated flow in place of wing and engine nacelle joint in Antonov 32 airplane was eliminated with vortex generators and wing fences. This led besides all to the increase of rate-of-climb by 60 fpm (0,3 m/s) in one-engine flight at take-off.

The installation of fence-type vortex generators on the engine nacelles of Antonov 140 airplane completely preclude flow separation phenomena in their tail parts.

Vortex generators are efficient means for task-oriented boundary layer control. At the same time airplane developers use this means with care, thoroughly weighing up obtained local aerodynamics enhancement (and improvement of airplane stability and control characteristics) with its energy-spending cost.

[1] Antonov Design Burean, Kyiv, Ukraine

Models for the Unsteady Surface Suction/Injection Flow Control

I.I. Lipatov[1]

The distributed or localyzed surface suction/injection could provide effective tool of the laminar boundary layer flow control to prevent separation or to delay laminar-turbulent transition. It was found experimentally (Seifert et al.) that unsteady mass transfer through the porous part of the profyle could diminish the extent of separation region and to increase critical angle of attack. The clear explanation of physical reasons leading to this effect is incomplete and additional analysis is needed.

Some aspects of this problem analysis is presented . It was supposed that the Reynolds number is large but don't exceed the critical value , therefore the boundary layer flow is supposed to be laminar.

Asymptotical methods (method of the matched asymptotic expansions) are used to analyze the problem. As a result similarity lows are deduced and corresponding mathematical problems are formulated. These problems are appropriate to describe nonlinear unsteady 2-D and 3-D flow control in the boundary layer flow. From mathematical point of view these problems are simpler that original Navier-Stokes model and more complex in comparison with the boundary layer model. Results of analytical solution of some linearized problems are presented. Prospects of the optimyzed flow control are discussed.

[1] *Central Aerohydrodynamic Institute (TsAGI), Zhukovsky-3, Moscow region , Russia*
ph: (095)-556-43-72, fax: (095) 911-00-19, e-mail: lipatow@postman.ru

Intensification of Near Wall Turbulent Flow Control with Aid of Vectored Suction (Blowing)

M.Lounis, V.T.Movchan[1]

Control of boundary layer by suction and blowing is well known, but in the past, this technology has not found a large practical application in aircraft building. The principle cause is the relativly lower gains in skin-friction drag reduction by delaying with aid of suction the laminar to turbulent transition of the boundary layer over streamlined aircraft surfaces (wings, engine nacelles, etc...) in addition to design, manufacturing and operation (maintenance) problems.

During few last years we observe a return of interest to this technology. Action of vectored suction and blow on streamlined surface is not studied as well as suction (blow) of boundary layer by normal to the wall, particularly for the case of turbulent flow. In the present communication were presented results of theoretical analysis of the influence of vectored uniforme porous suction (blow) on skin-friction drag, heat transfer and near-wall turbulent flow control.

In this work, considered principle parameters of suction (blow) are angle between porous mass transfer velocity vector and the normal to the wall, witch is varied from 0 deg. (fully normal action) to 90 deg. (fully tangential action) and velocity module of suction (blow) related to basic flow velocity. To resolve partial differential equations system a develloped by authors algebraic model for turbulent viscosity and thermal conductivity and validated for computations of a large class of near-wall turbulent flows was used.

Numerical testing of the method for the case of fully develloped turbulent pipe flow gave results, witch demonstrate that, due to variations of vorticity structure inside flow near-wall region the basic flow parameters like pressure gradient, tangential stress, velocity components and temperature profiles, skin-friction and heat transfer coefficients sensibly depend on both considered parameters of suction and blow.

In the cases of fully normal suction and blow skin-friction coefficient is higher and lower respectivly. When the angle of suction rises, the skin-friction coefficient drops. At 45 deg. it equals to value relative to non-porous wall and at 90 deg. we obtain the minimal value. It means that at angles higher than 45 deg. suction gives possibility to reduce significantly the friction drag of turbulent flow. The presence of tangential suction component induces reduction of wall shear stress and minimises influence of the normal suction component, effects of witch are opposite. Effect of vectored blowing is positive at all angles (friction reduction) and have non-monoton form with minimum value of skin-fiction coefficient at 45 deg.

As well as the results were obtained for pipe flow they can't be extrapolated for the case of classic boundary layer. Otherwise, this analysis offers a preleminary evaluation of vectored suction (blow) effectiveness in application with aim to resolve relative to aerospace engineering several important problems like intensification of turbulent boundary layer control, heat transfer and skin-friction drag reduction in both outer and inner viscous flows.

[1] *Kiev International University of Civil Aviation, Department of Higher Mathematics, (Building #8), 1, Cosmonaut Komarov Avenue, Kiev - 058, Ukraine.* *Tel.: (38044) 484 92 14*

Ten/Hundred Time Seawater Drag Reduction as a Real Problem of XXI Century

V.I. Merkulov[1]

In the present work it is shown that the periodic flow is suitable for reduction of the friction drag to zero. To maintain such flow, a work of the order 1/Re must be performed in the liquid. Thus, the radical multiple reduction of hydrodynamical drag can be achieved at the expense of transition to periodic flow. As the first example of such flow, motion of a toroidal body inside a ring vortex is considered. It is known that the external flow around a ring vortex is potential and does not create drag, and the internal flow is periodic. As applied to motion in seawater, constant electrical and magnetic fields are used for compensation of viscous losses caused by internal vortical flow. The required capacity has the order of 1/Re.

As calculations have shown, at magnetic force in one tesla generated by using of constant magnets, power of 300 watt is necessary to move a torus of 2 meters in diameter at speed of 10 m/s.

This power is four orders less than on-coming flow power. Electrical efficiency at the specified parameters is equal to 6 percents.

The electromagnetic fields allow creating and supporting periodic flow along an infinite plate. Thus the Reynolds number, based on the wavelength, has rather small subcritical value, but is large enough to manage with weak magnetic fields.

The required power has the same order, as in the example with a toroidal body.

One more way to create a periodic flow is a traveling surface wave. There are both theoretical and experimental researches for such flows. Rostrum of a swordfish, due to its roughness, is a generator of spiral vortices that reorganize the boundary layer over the fishes to some periodic flow, with all subsequences.

[1] Institute of Theoretical and Applied Mechanics, SB of the Russian Academy of Sciences, Novosibirsk, 630090, Russia
E-mail<vimer@mail.nsk.ru>

Mathematical Modelling of Turbulent Wall Jets on a Rough Streamlined Surface

V.T.Movchan, E.A.Shkvar[1]

Wall jets are known as a efficient method of active boundary layer control with the aim to prevent its separation. On the another side, wall jet has the negative effect of skin friction increasing. In connection with this the factor of wall roughness must be considered as one of basic parameters, which determines the properties of near-wall jet shear flow. The interaction between tangential near-wall blowing and wall roughness produce the complicated structure of vorticity and, as a result, turbulent viscosity distribution across the boundary layer thickness. This problem must be accounted in mathematical model of this type of flow.

The aim of this work is to show the results, which were obtained by authors in the direction of modelling and prediction of this kind of viscous flows.

Mathematical model is based on boundary layer type system of equations, which include continuity equation, longitudinal impulse transfer equation. With the aim of closure of this system the algebraic model of turbulence in the form, which was proposed by prof. Movchan V.T., has been used. The modification of this model by using functions of shift of logarithmic zone of velocity profile, which was proposed by Shkvar E.A., gave us the possibility to account the influence of roughness of streamlined surface.

As an alternative way to algebraic approach of turbulence modelling the k-ε model of turbulence was modified under mentioned above circumstances. The advantages and disadvantages of these methods of turbulence characteristics predictions will be discuss and analyzed.

For solving partial differential equations system the effective marching method with second order accuracy has been elaborated. The general advantage of proposed finite-difference scheme and computational method is without-iteration two-step procedure, which allows quickly running along flow direction.

The numerical experiment shows the workability of proposed hypotheses and effectiveness of proposed computational method of boundary layer and semi-empirical model of turbulence in the case of the presence both tangential blowing and wall roughness.

The obtained results can find application as the tool of optimization of a construction and justified choice of regimes parameters of an equipment developed with the aim of shear flows control by using tangential wall jet.

(Higher Mathematics Department, Kyiv International University of Civil Aviation, Komarov ave., Bld. 1, 02058, Kyiv, Ukraine, e-mail: shkvar@i.com.ua)

Use of Vortical Streams in Time of Treatment of Metal Melts out of Aggregate

V.L.Naydek, G.F.Petrov[1]

The assurance of effective stirring or turbulization of respondent phases in processes of treatment of melts is very important problem in metallurgy. As a rule, energy is expenditured on this additionally.

There were researches of hydrodynamic peculiarities of vortical streams were conducted in PTIMA of NAS of Ukraine and it was given an estimation of possibility of use of vortical devices for intensification of mass exchange and decrease of resources expenditures. The vortical stream, which expires, from crater inherits acquiring rotatory movement of melt and axial break of compactness of flow on the sizeable stretch.

On the base of results of modeling we determined presence of region of automodel currents by defined criterions of Reunolds and Weber (fig.1,b) and registered that steady work of vortical device were assured in wide range of value consumption, which satisfying correlation $0,6Q_{max} < Q < Q_{max}$. This is testifying about great possibilities in construction of such devices. Varying of consumption of melt by equipment and geometric parameters of vortical device allows changing geometry of hollow part of whirlwind. This changes drawing forces and limiting consumption of putting addition. In comparison with continuous incident stream degree of turbulization of melt expressed by Reunolds's number increases in 15-20 times in his admission through vortical device. Time of contact between additions and melt rises on order.

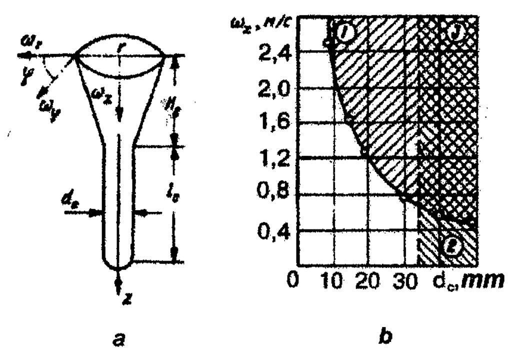


Fig.1. Scheme of model of vortical stream (a), region of automodel of current of liquid (b):
1 - by Reunolds's criterion;
2 - by Weber's criterion;
3 - by Reunolds's and Weber's criteria.

Experimental data are presented at fig.2, which characterize capacity of vortical stream to transport a gas phase from camera cavity of twist of vortical device at different expense characteristics of jet. The changing of consumption of liquid doesn't influence on mechanism of formation and structure of zone of interaction between stream and bath. Only quantity of ejected air and depth of aeration zone under practice constant its diameter is subjected to changing.

During researches it was established that two principal different variants melts of out of furnace treatment can be realize by additions with use vortical device: BA-1 with pouring nozzle disposed on considerable distance from surface of melt (h>>0) and BA-2 with nozzle is contiguous with surface and is deepened into melt ($h \le 0$).

[1] *Physics-technological institute of metals and alloys of NAS of Ukraine (Ukraine, Kiev)*

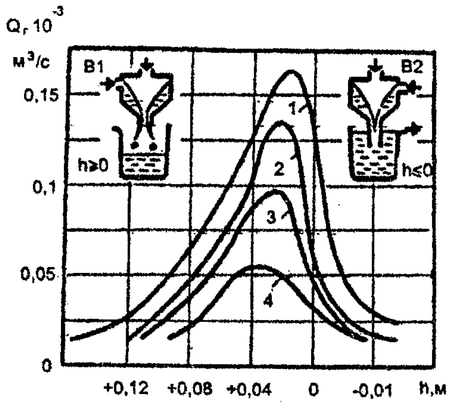


Fig.2. The dependence of consumption of ejected gas in vortical device on position of nozzle regarding to surface of melt at different consumption of smelt.

In first variant of vortical device the kinetic energy is used for crushing of stream with aim of efficiency rise of treatment of melt by reagents in drop refining of metal.

In second variant of vortical device (BA-2) the pouring nozzle situate in the contact with bath of melts. It lets to implement its treatment by evaporating and high active reagents in gas medium of the necessary composition.

Researches had determined that from 80 to 90% of potential energy is transformed in kinetic energy of vortical stream and is dissipated by melt at use in vortical devices of long nozzles in case their contact with bath. In this case the kinetic energy of stream is used on mixing polyphase system transported by vortical stream in bath in full measure. Necessarily the energy of mixing can be increase due to forced feed of gas in crater and combined influence of melt of vortical and gas jets on the bath.

The calculation of power of mixing of melt was conducted for realization of comparative analysis of efficiency of different methods of hydrodynamic treatment of metals in the same conditions for free falling jet and jet of metal is poured through vortical devices of different constructions.

From point of view of intensification of mass exchange process more effective is vortical device BA-2, in which the nozzle contacting with bath of smelt and inert or other gas is blown through device compulsorily.

Using developed procedure of calculation a number of vortical devices of different productivity was manufactured in PTIMA of NAS of Ukraine. Their tests were carried out in laboratory and industrial conditions in treatment by refined additions of cast iron smelted in cupolas, induction and domain furnaces. The degree of removal of sulfur from melt reached to 70-75 % at that.

On the Stability of Turbulent Shear Flow

Bernd R. Noack[1], Fabio P. Bertolotti[2]

The stability property of time-averaged turbulent flow is theoretically and numerically investigated. Linear stability analysis for time-averaged turbulent flows is mathematically shown to describe the growth of an infinitesimal perturbation in unsteady flow under weak conditions for the turbulence spectrum. Turbulent flow is analytically shown to be marginally stable under well-defined mathematical assumptions. The marginal stability property is corroborated by a numerical stability analysis of plane Poiseuille flow, of boundary layers and of shear layers. The result is of relevance for the turbulence modeling, in particular for identifying potentially non-physical solutions and for a priori adjustment of a free parameter.

1. Introduction

The stability property of time-averaged turbulent flow can have important implications for turbulence models. Prandtl (1945) outlined that the eddy-viscosity ansatz employed in many present turbulence models neglects potentially important stability properties of the flow. Malkus (1956) conjectures on qualitative grounds that time-averaged flows can be expected to be marginally stable. This conjecture is employed in several turbulence models, for instance in the derivation of far-fields asymptotic of shear flow (Lesson &Singh 1974; Lesson &Paillet 1976, Lesson 1978) and in Stull's (1993) transilient turbulence theory for atmospheric boundary layers. In the present study, marginal stability is for the first time analytically derived under well-defined mathematical assumptions. In addition, numerical evidence is presented for a variety of different flows.

2. Theoretical model

Incompressible, viscous flow is considered in a finite domain with steady Dirichlet boundary condition or periodic boundary conditions. These conditions apply to confined flows and fields restricted to a periodically continued box as in simulations for Poiseuille flow. The turbulent flow is modeled by a triple-decomposition ansatz (Reynolds & Hussain 1972; Liu 1989)

$$\mathbf{u} = \bar{\mathbf{u}} + \mathbf{u}_1 + \mathbf{u}_2,$$

where $\mathbf{u}$ denotes the velocity field. Time-averaged quantities are denoted by a bar and the subscripts, 1,2, refer to the periodic coherent-structure and the stochastic turbulent contribution, respectively. The periodic contribution is modeled by the normal-mode ansatz $\mathbf{u}_1 = A\Re\{\mathbf{f}(\mathbf{x})e^{i\omega t}\}$ where $\mathbf{f}$ is the most amplified complex eigenvector of the time-averaged flow. The second contribution $\mathbf{u}_2$ is assumed to be statistically independent from $\mathbf{u}_1$. Additionally, the most amplified infinitesimal disturbance of the time-averaged flow, is described by $\mathbf{u}^* = A\Re\{\mathbf{f}(\mathbf{x})e^{\sigma t + i\omega t}\}$ with growth-rate σ. Derivations starting with the unsteady Navier-Stokes equations yield a vanishing growth-rate $\sigma = 0$ without further assumptions.

3. Numerical study

It has been a common view in the past that a 'complete' and 'correct' linearized stability analysis of a turbulent flow, $\mathbf{u}$, should include the interaction between the infinitesimal disturbance, $\mathbf{u}^*$ and the stochastic part of the turbulence field, $\mathbf{u}_2$ and concern has been raised that the nature of this interaction is not known. Using an alternative, but complementary, mathematical method to that used in our proof of marginal stability, we show that in strictly parallel flows, the interaction between the stability mode of infinitesimal amplitude and the turbulence field is negligible, and in weakly non-parallel

1 *Hermann-Föttinger-Institut für Strömungsmechanik, Technische Universität Berlin, Strasse des 17. Juni 135, D-10623 Berlin, Germany; Phone: ++49 30 314-23359; Fax: ++49 30 314-21101; E-mail: noackbr@pi.tu-berlin.de*

2 *United Technologies Research Center, 411 Silver Lane MS 129-15, East Hartford, CT 06108; Phone: ++1 860 610-7224; Fax: ++1 860 610-2108; E-mail: BertolFP@utrc.utc.com*

flows the interaction is weak, so that the analysis based on only the averaged turbulent profile is meaningful. Our numerical results consist mainly of eigenvalue spectra for available DNS results for turbulent flows in channels, boundary-layers, and free shear layers, and spectra from RANS solutions. These results extend the analysis of Friedrich and Bertolotti (1997) to a variety of computed flows. The numerical results tend to support the marginal stability hypothesis. Effects of the Reynolds number and the numerical resolution in the DNS simulation are discussed, as well as the effect of 'non-equilibrium' in the turbulence field.

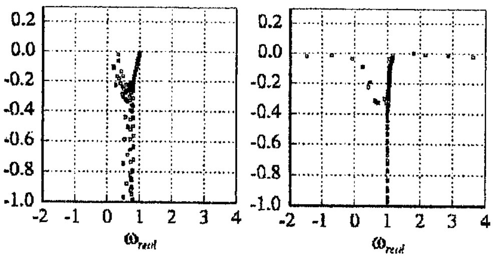


Figure 1: Stability spectra of a local stability analysis. The analysis is based on profiles from direct numerical channel flow simulations at Re=4880. Left: incompressible unsteady laminar-periodic flow. Right: turbulent, compressible flow at Ma=3. Note the spectra touch the line of neutral stability, i.e. vanishing ordinate value.

4. Conclusions

For the first time, Malkus's (1956) principle of marginal stability is rigorously derived under plausible mathematical assumptions for a larger class of flows. Thus, deviations from marginal stability can be traced back to a violation of the assumptions, i.e. the normal-mode ansatz and the statistical independence of coherent and stochastic contribution. Numerical investigation indicates that the equilibrium flow satisfy well the marginal stability property. Converging (diverging) flows with beneficial (adverse) pressure gradients tend to be more stable (unstable).

The marginal stability property can be employed as a plausibility check for suggested flow solutions of turbulence models. A strong instability of a suggested flow from a steady turbulence model may indicate that an unsteady version of the model needs to be employed. In addition, free parameters may be calibrated using this property.

The authors acknowledge stimulating discussions with K. Afanasiev, S. Leibovich, M. Morzyński and F. Thiele. The work is partially supported by the Deutsche Forschungsgemeinschaft (DFG) under Grant No258/1-1.

Bibliography

Lessen, M. & Singh, P.J. Stabil(1974): ity of turbulent jets and wakes. *Phys. Fluids 17, 1329-1330*

Lessen, M. & Paillet, F. *(1976):* Marginal instability of turbulent shearing layers and the break point of a jet. *Phys. Fluids 19, 943-944.*

Lessen, M. *(1978):* On the power laws for turbulent jets, wakes and shearing layers and their relationship to the principle of marginal instability. *J. Fluid Mech. 88, 535-540.*

Malkus, W.V.R. *(1956):* Outline of a theory of turbulent shear flow. *J. Fluid Mech. 1, 521-539.*

Prandtl, L. *(1945):* Bericht über ein neues Formelsystem für die ausgebildete Turbulenz. Nachrichten von der Akademie der Wissenschaft in Göttingen, *Mathematisch-Physikalische Klasse.*

Reynolds, W.C. & A.K.M.F. Hussain (1972): The mechanics of an organized wave in turbulent shear flow. Part 3. Theoretical model and comparisons with experiments. *J. Fluid Mech. 54, 263-288.*

Stull, R.B. *(1993)*: Review of non-local mixing in turbulent atmospheres: transilient turbulence theory. *Boundary-Layer Meteorology 62, 21-96.*

Friedrich, R & Bertolotti, F. P. (1997): Compressibility effects due to turbulent fluctuations. *Applied Scientific Research 57, 165-194.*

Polymer Macromolecules as a Tool For Studying Under Wall-Adjacent Turbulence Flow

Volodymyr Pogrebnyak[1]

The interest of specialists in hydrodynamics in the flow of solutions of polymers revealing low drag in the mode of turbulence is explained not only due to prospects of practical application of soluble polymers but also due to aspiration to use polymer additions as a tool to provide more complete study of turbulent flows.

Among the attempts to explain the nature of effect Toms', lying in drag reduction by the polymeric components, special place is taken by a hypothesis, based on strong deformation effect of a near-the-wall turbulence on macromolecules. For the substantiation of this hypothesis experimental proofs of presence of large degrees of deformation of macromolecules in a wall-adjacent zone of a turbulent flow are necessary. The skepticism concerning strong deformation effect of wall-adjacent turbulence on macromolecules is stipulated yet by the fact that, as a rule, shift effects wall-adjacent a turbulence are analysed, and not the, jet flows ("explosions") with a longitudinal gradient of speed which arise in the wall-adjacent area.

Therefore the experiments proving the stretching of molecules in conditions of wall-adjacent turbulence have a fundamental character not only in point of developing the mechanism of drag reduction by polymer additions but also in point of more profound insight into the nature of turbulence itself.

Polyethylene oxide (PEO) molecules have their own anisotropy and anisotropy of the form. And in case of full deployment macromolecules, which can be realized only in streams with expansion there, happens very significant (by 3-4 orders of magnitude more, than for simple shift) and fast increment of factor of birefringence with the increase of a gradient of velocity acting on molecular coils, up to extremely possible Δn_{∞}, (D'yakova and et. al., 1989).

PEO having the viscosity-average molecular weight of $\overline{M}_{\eta} = 4.10^{6}$ was used as a polymeric additive and latent root of a viscosity 1,72 m^3/kg . The solutions of polymer had concentration 0,01-0,05% . Special hydrodynamic canal a rectangular cross-section 4x4 mm and height of 1m was used. Visualization made in wall-aljacent of area located on a distance 0,25m from its extremity of the channel .

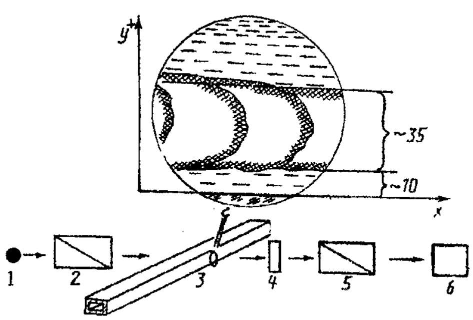


Fig.1. The sketch of a picture of birefringence zone and a diagram of experimental installation (1-mercury lamp; 2-polarizer; 3-hydrodynamic channel; 4-plate a equal to a quarter of a wavelength; 5-analyzer; 6-camera or cine camera).

Fig.1 shows a ketch of a picture birefringence zone of a turbulent boundary layer of a current of a water solution PEO having concentration 0,05 % and Re=2 $\cdot 10^4$. Polarization and optical visualization testify, that in the field of $10 \le y+ \le 45$, in which maximum generation of jets of fluid happens, birefringence is localized. In the immediate proximity to the wall ($0 \le y^+ < 10$) and in the field of the

[1] Ecological Center of scientific and applied researches
31, Shorsa Street, Donetsk, 83050, Ukraine
E-mail: VP@ecocenter.dn.ua

kernel (y^+>45) birefringence is unavariable ($y^+ = V \cdot y/\nu$, where: y- is coordinate; V- is dynamic velocity; ν- is cinematic viscosity). This fundamental experiment unequivocally shows the presence of strong deformation effect on the molecular coils in certain zones of a turbulent boundary layer, having supercritical longitudinal gradients of speeds. A mean angle of the orientation of polymer molecules with a direction of the main flow is ~35°. The performed visualization of near-the-wall turbulent flow of water solution PEO has confirmed the conventional ideas about the structure of a turbulent boundary layer.

The adequate experimental confirmation of the unrolling of macromolecules in conditions of the near-the-wall turbulence has made obvious the advantages of using nonturbulence flows with stretching for researching interaction of macromolecules with a hydrodynamic field and allowed to model the main structural singularities of a turbulent boundary layer.

Velocity and velocity gradient fields arising at the entrance of a small capillary during the free-converging flow (in conditions of near-the-wall turbulence) of aqueous solution of polyethyleneoxide, as well as the degree of the coil-stretch transition of the macromolecule were experimentally studied. The hydrodynamic field arising under the converging flow conditions resulted in a considerable (~60%) degree of polymer stretching which, in turn, led to a readjustment of the hydrodynamic field itself.

The comparison of the data, obtained when modelling eruptions of microjets of a polymeric solution from the near-the-wall area (Pogrebnyak and et.al.,1984) and frequency harmonic effect of solutions PEO on the flow with stretching (Pogrebnyak,2000), with known results on the anisotropy of speed fluctuations and suppression of their high-frequency part, completely agree between themselves (Kutateladze and et.al.,1975), i.e. turbulent (in macroscopic scale) the current is perceived as laminar (in microscale) in case of interaction of a hydrodynamic field with molecules of polymer.

With the help of direct experiments it has been proved, that in halfdiluted and moderately concentrated flooded polymeric jets there is observed the dynamic structure generation under the action of supercritical longitudinal gradients of speed (Pogrebnyak,1992). The research of high-speed converging flows (~250m/s) testify to certain role played by the longitudinal gradient of speed realizable in case of the flow in deforming macromolecular coils and minor role of a transverse gradient of speed (Pogrebnyak,1995).

Relying on the data characterising the dynamics of macromulecules in nonturbulent flows with stretching, as well as on the proven strong deformation effect of wall-adjacent turbulence on macromolecules, and also using the data of model researches of singularities of turbulence in a boundary layer (see the above and Pogrebnyak and et.al.,1984), we have offered molecular-and-supermolecular mechanism of the effect of flow resistance drop when introducing of the soluble polymeric components in a turbulent flow. The mechanism of Toms effect consists in the appearance of a self-oscillating mode of reversible processes of unrolling macromolecules under the action of quasiregular longitudinal gradients of speed in a turbulent boundary layer as well as influence teared macromolecules both on molecular (at $C < C_{opt}$), and on supermolecular (at $C > C_{opt}$) levels on the structure of near-the-wall turbulence, i.e. As a result of oscillations of deformations of macromolecules and dissolution of dynamic supermolecular formations, arising under the operation of stretching flows. All this results in the increase of the period of jets of fluid into the external part of a boundary layer and, as a result this causes the thickening of viscous underlayer. As a result of it generation of primary turbulence decreases and general level of turbulent dissipation in the flow gets reduced. In case of too large molecular masses and increase of viscosity above some limit of concentration, stipulated both by the "usual" intermolecular interaction and by the ynamic structures, under formation results in sharp decrease of Toms' effect.

Numerical Studies of Heat and Mass Transfer Effects on Formation, Development and Separation of Turbulent Boundary Layers

O.A. Prykhodko, O.B. Polevoy[1]

The problems of the efficiency increase of gas turbo plants, heat-transfer-units, optimization of airodynamical shapes attach a special interest to investigations of heat and mass transfer influence on development and characteristics of supersonic turbulent separated flows.In presented paper the influence of cooling and heating of streamlined surface, free mass transfer, isothermal distributed injection and suction on development separated turbulent flows has been considered.The investigations have been carried out numerically on the base of averaged Navier-Stokes equations, completed by turbulent viscosity model, for fixed parameters of incominig flow under different pressure overfalls, defined by the angle of stream turn behind the shock wave or by the off-design parameter of jet for concentrated injection. The changings of general structure of viscid-inviscid separated interaction, distributions of pressure, skin-friction and heat transfer coefficients are analysed. The dependences of basic characteristics of supersonic turbulent separated flows from determined parameters of heat and mass transfer and pressure gradient are presented. The obtained results are compared with known experimental data and other authors' computations. It is showed, that in spite of the different phisical basis of phenomenon, the changings of the conditions of heat and mass transfer possess a similar risult influence on structure and characteristics of supersonic turbulent separated flows.

[1] *Dnepropetrovsk State University, UKRAINE*

Passive and Active Control of Separation

R. B. Rivir[1]

Abstract

Passive and active control of separated flows has been demonstrated successfully by a number of techniques which employ the introduction of longitudinal or parallel to the primary flow vortices or combinations of both. The role of these vorticies is basically to reenergize the wall boundary layer flow by entraining and redistributing momentum from the primary flow to the wall layer. These vortices also prevent and inhibit span wise break down further reducing mixing losses. The passive generation techniques include half delta wings or fences currently extensively employed on external aircraft flows, riblets employed primarily on external flows, dimples employed on golf balls, roughness in the form of trip strips or "micro dots". Active techniques include passive or actively forced flaps, suction or blowing, thermal riblets, synthetic jets, surface deformation, electrostatic and plasma interactions with flows, acoustic cavities or forcing, electromagnetic flow interactions, and mems devices employing various combinations of the previous techniques. Examples of these devices will be illustrated with some of their characteristics. Dimples and pulsed 90o skew injection with a duty cycle down to 1% have nearly eliminated the separation losses of a low pressure turbine cascade operating at Reynolds numbers down to 50,000, with out incurring additional losses at higher Reynolds numbers. The dimples and pulsed injection experiments will be described in detail.

[1]*Air Force Research Laboratory, Propulsion Directorate, Wright-Patterson AFB, OH 45433-72251*
e-mail : richard.rivir@wpafb.af.mil

Control of Transition in 3-D Boundary Layers with Streamwise Vortices in the Presence of Freestream Turbulence and Random Surface Roughness

William S. Saric[1], Robert Gladden[1], Pierre Gabet[1] and Edward B. White[2]

Introduction

Crossflow-dominated transition on a swept wing represents both a fundamentally challenging and a technologically important research problem. The nature of the crossflow instability is such that the boundary layer is subject to strongly nonlinear behavior very early in its evolution.[1] Because of this, it has resisted treatment by linear methods and predicting the transition location for even the simplest configurations is not presently possible. This is in spite of the fact that a very complete understanding of the primary crossflow instability has been developed and very good agreement between experiments and computations has been obtained. What is lacking is a detailed *experimental* description of the secondary instability, the process by which the boundary layer structures that result from the nonlinear primary instability break down and result in turbulent flow. The secondary instability process has been investigated experimentally[2] and a computational model for it exists,[3] but it is not clear that these results are in agreement. It is the objective of the current work to provide data on the breakdown region of a stationary-wave-dominated crossflow boundary and to extend the earlier transition control work of Saric et al.[1] to the case of increased background surface roughness and increased freestream turbulence levels.

1. Experimental Approach and Results

The present experiment consists of making detailed measurements of the mean and fluctuating velocity components in the boundary layer of a 45° swept wing in the ASU Unsteady Wind Tunnel, a low-disturbance facility specifically designed for transition-to-turbulence experiments. The model is a natural laminar flow design. The pressure minimum is at 71% chord so only the crossflow instability is active in the test region. Arrays of 18- or 54-μm-high roughness elements on a 12-mm spacing just aft of the attachment line fix a stationary primary instability pattern that saturates and provides the basic state for secondary instability measurements. Figure 1 is a typical mean-flow contour just prior to breakdown. This contour shows inflection points in the streamwise velocity both in the wall-normal (Y) and spanwise (z) directions. High-frequency secondary-instability fluctuations such as those in Figure 2 reveal that it is usually the spanwise inflection point that supports the dominant secondary instability mode. In some instances however, the wall-normal inflection point supports a secondary instability mode as well. In every case examined, the secondary instability grows very rapidly just prior to breakdown, and the location of breakdown can be correlated to the location of maximum secondary instability activity. These results confirm the computational results of Malik et al.[3]

Preliminary experiments with painted surfaces demonstrate that the control mechanism of Saric et al.[1] with subcritically-spaced roughness still works. A distribution of 50-μm-high roughness elements at a 8 mm spacing stabilized the boundary layer beyond the pressure minimum even though the surface had paint-induced random roughness of 11-30-μm. When the freestream turbulence levels were increased from 0.02% to 0.3%, transition was dominated by travelling crossflow waves and subcritical roughness lowered the transition Reynolds number. This raises questions regarding the utility of these experiments in relatively high turbulence levels characteristic of all but a few wind tunnels.

[1] Arizona State University, Tempe, AZ 85287-6106, USA
[2] Case Western Reserve University, Cleveland, OH 44106-7222,USA

Acknowledgement/Disclaimer

This work was sponsored (in part) by the Air Force Office of Scientific Research, USAF, under grant/contract number F49620-97-1-0520. The views and conclusions contained herein are those of the authors and should not be interpreted as necessarily representing the official policies or endorsements, either expressed or implied, of the Air Force Office of Scientific Research or the U.S. Government.

References

1) Saric, W.S., R.B. Carrillo, Jr., and M.S. Reibert. "Nonlinear stability and transition in 3-D boundary layers." *Meccanica*, **33**:469-487 (1998).

2) Kohama, Y., W.S. Saric, and J.A. Hoos. "A high-frequency, secondary instability of crossflow vortices that leads to transition." Proc. Royal Aero. Soc. Conf. On Boundary-Layer Transition and Control (1991).

3) Malik, M.R., F. Li, and C.-L. Chang. "Crossflow disturbances in three-dimensional boundary layers: Nonlinear development, wave interaction and secondary instability." *J. Fluid Mech.*, **268**: (1994).

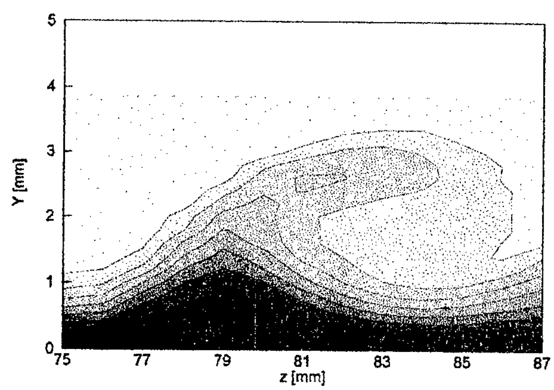


Figure 1. Mean-flow contours for Re_c=2.4×10^6, x/c=0.45, [18|12] roughness. Contours are 10% of u/u_e.

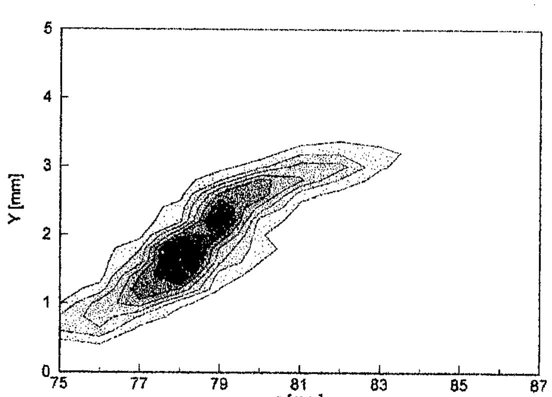


Figure 2. RMS contours of the secondary instability mode shape (2.9-3.1 kHz passband) for Figure 1. Lines are 10% contours of the maximum amplitude.

High Molecular Weight Polymeric and Micella-Constitutive Surface-Active Microadditives Applying for the Hydrodynamic Drag Force Turbulent Fluid Flow Control

A.B.Stupin[1], P.V.Aslanov[2], A.P.Simonenko[3], S.A.Fomenko[4], N.V.Bykovskaya[5]

In many manufacturing processes and kettles of the modern industry the streams of the different fluids (the service water will widely be utilized, the warm and cold supply etc.). For the diminution of power inputs on the pumping-over of fluid mediums, and by that pinches of an overall performance of technological devices and systems, the targeted action on the performances of a fluid fluxions permitting to achieve essential diminution of turbulent friction is necessary. Perspective in this direction is a usage of different miscible (of the high molecular weight polymers, micella-constitutive surface-active additives - MSAA) and nonsoluble additives is, which one can essentially (up to 80 %) slash hydrodynamic drag force of friction at a motion turbulent mode [1-6, 9, 10].

The results of a comprehensive analysis of turbulent flows of surfactant solutions are presented. A laser Doppler velocimeter was used to determine the mean velocity and turbulence intensity profiles, the turbulent stresses and the energy spectra. The experimental data were used to calculate the anisotropy of velocity fluctuations, their skewness and kurtosis, and the component of the energy balance of the flow, namely turbulent energy generation, dissipation of energy of the average flow, and stored elastic energy. For comparison, certain analogous characteristics were measured or calculated for flow of a polymer (polyacrylamide) solution.

The above investigations enable us to identify common and distinctive characteristics of the effect of surfactant and polymer additives on wall turbulence.

The common features in the flow of surfactant and polymer solutions (compared with water) are: an increase in the thickness of the viscous sublayer, the generation of elastic stresses and the presence of stored elastic energy, a decrease in turbulent drag, and an increase in the skewness of the probability distribution of longitudinal velocity fluctuations near the wall.

The points of difference are: much lower intensity of longitudinal velocity fluctuations in surfactant solutions than in polymer solutions, and much lower anisotropy of turbulent velocity fluctuations in flows containing surfactants.

To interpret these characteristics, we consider the possible physical mechanics that produce elasticity and decrease the turbulent drag in flows containing additives.

Polymer Solutions. We know that in a flow with transverse shear the degree of ileformation of the macromolecules is very small: under real conditions the extension on the polymer molecules will be significantly high only in extensional flow [3, 6]. The interpretation of the physical mechanism of this type of flow is based on a model of turbulence with extension (stretching) of vortex tubes [7].

When polymer molecules are present in a turbulent flow, they interact with vortex tubes by the following mechanism. In the presence of fairly high shear, most of the polymer molecules will at any instant be oriented at an angle on that is close to zero. Therefore, the macromolecules that have the highest probability of being trapped in a vortex tube are those whose axes are oriented in the direction of flow. Part of the energy of the vortex tubes will then be expended on stretching the polymer chains, and thus will be stored as elastic energy.

In fluctuating turbulent flow, the liquid with partially or completely oriented macromolecules will exhibit anisotropic viscosity [3, 6]. This is because in the presence of oriented polymer molecules, the different directions in the flow are not equiprobable for occurrence of fluctuating motion. The elastic polymer chains will affect primarily transverse velocity fluctuations, which they will damp out.

Overall, this approach gives a physical explanation of the main characteristics of turbulent flow in polymer solutions: the generation of elastic stresses and the existence of stored energy; the decrease in the energy of transverse velocity fluctuations and turbulent drag, and the increase in the anisotropy of turbulent fluctuations. The hypothesis of anisotropic viscosity also provides an explanation for the increase in thickness of the viscous sublayer in flow of polymer solutions.

[1,2,3,4,5] *Donetsk State University*
(Universitetskaya str., 24, Donetsk, Ukraine, 83055; E-mail: stupin@dongu.donetsk.ua)

Dilute Surfactant Solutions. The physicochemical properties of surfactant solutions and the generation of viscoelasticity in them are described in detail in [8], whose authors conclude that a periodic colloidal structure (PCS) with a quasicrystalline lattice must arise in a colloidal solution containing cylindrical micelles in order for viscoelastic properties to occur. Local PCS's form at relatively low surfactant concentrations.

The concept of the PCS can be used to explain our experimental data on the turbulent structure of flows of surfactant solutions. We believe chat the generation of elastic stresses and the existence of stored elastic energy in the flow of surfactant solutions result from deformation of the quasicrystalline lattices of PCS's in shear flow. Since local PCS's have only slight skewness, the effect of dynamic forces in the flow will be simply to deform the structures without a pronounced orienting effect. As a result, the elastic PCS's will have a damping effect on both transverse and longitudinal components of the fluctuating velocity. The result of simultaneous damping of these components will be that the anisotropy of turbulent velocity fluctuations is much smaller in surfactant solutions than in polymer solutions.

Thus our analysis of the common features of flow of surfactant and polymer solutions indicates that the decrease in turbulent drag results from the generation of viscoelasticity, which ultimately results in damping of turbulence. The points of difference between dilute surfactant and polymer flows result from the particular ways in which elasticity is generated in these liquids: i.e., in polymer solutions as a result of stretching of macromolecular chains, and in surfactant solutions as a result of deformation of the quasicrystalline lattices of local PCS's.

In the report the results of systematization of the additives, lowering turbulent friction, compositions on their basis are given also, the technologies of their preparing both applying for interior and exterior problems irradiated, the basic legitimacies of a resistance drop in tubes and channels are introduced [9, 10].

References:

1. **Povkh I.L., Stupin A.B., et al**. *Turbulent Flow of Solutions of Micelle-Forming Surfactants. In: Mekhanika turbulentnykh potokov (Mechanics of Turbulent Flows), Moscow, 1980, 44-69.*
2. **Aslanov P.V., Maksyutenko S.N., Povkh, I.L., Simonenko, A.P. and Stupin, A.B.** *Turbulent Flow of Surfactant Solutions. Izv. Akad. Nauk SSSR. Mekhanika zhidkosti i gaza, no. 1, 1980, 36-43.*
3. **Pilipenko V.N.** *Effect of Additives on Turbulent Wall Flows. Itogi nauki i tekhniki. Mekhanika zhidkosti i gaza (Progress in Science and Engineering-Fluid Mechanics), 15, VINITI Press, Moscow, 1980, 156-257.*
4. **Stupin A.B., Aslanov P.V.** *Turbulent Structure of Flows with Drag-Reducing Additives. Inzh.-fiz. zhu., 41, no. 5, 1981, 809-814.*
5. **Povkh I.L., Stupin A.B., Aslanov P.V.** *Structure of turbulence in flows with surfactant and polymeric additives. Fluid Mechanics-Soviet Research, 17, no. 1, 1988, 65-79.*
6. **Stupin A.B.** *Semiempirical theory of drag reduction. Fluid Mechanics-Soviet Research. 19, no. 5, 1990, 104-114.*
7. **Frost W. and Bitte J.** *Statistical Concepts in the Theory of Turbulence. Russian translation in: Turbulentnost: Printsipy i primeneniya, Mir Press, Moscow, 1980.*
8. **Ulmius J., Wennerstrom H., Johansson L., Lindblom G. and Gravsholt S.** *Viscoeiasticity in Surfactant Solutions. Characteristics of the Micellar Aggregates and the Formation of Periodic Colloidal Structures. J. Phys., Chem., **83**, No. 17, 1979.*
9. **Stupin A.B., Simonenko A.P., Aslanov P.V.** *Hydrodynamic-active compositions in energy supply and ecology, Donetsk State University, 1999, 230.*
10. **Stupin A.B., Simonenko A.P., Aslanov P.V., Bykovskaya N.V.** *The application of hydrodynamic active additives of polymers and surfactant substances in energy-saving technology. Proceedings of the 11th European Drag Reduction Working Meeting. Prague, Czech Republic. 1999, 46-47.*

3D Turbulent Boundary Layer Model Based on Organized Motion Theory

A. P. Trunev[1]

The model of the turbulent boundary layer in pressure gradient is considered in the paper. The model is based on the special type of the Navier-Stokes equation transformation and contains only molecular viscous parameters. The structure of eddy viscosity is determined from the turbulence theory. The pressure gradient and the random parameters characterising the roughness density and geometry are taken into account in the model.

The main idea of the special type of the Navier-Stokes equation transformation is to introduce the random parameters into the hydrodynamic equations. This is possible in a case of boundary layer, for which a surface layer transformation is performed, i.e. presentation of a flow velocity vector, $\mathbf{u}=(u,v,w)$, as $\mathbf{u}=\mathbf{u}(x,y,z/h(x,y.t),t)$, where z is the space variable which is normal to the wall, $h=h(x,y,t)$ - is the dynamic roughness (the surface in the area of flow, adjoining a solid surface but not coinciding with it). The dynamic roughness surface is tightly connected to the coherent structures in a turbulent boundary layer.

The random variable of velocity is defined by toting of the expression $\mathbf{u}=\mathbf{u}(x,y,z/h(x,y.t),t)$ in volume dV_s:

$$\tilde{\mathbf{u}}(\eta,t,h,h_x,h_y,h_t)=\lim_{\delta V\to dV_s}\frac{1}{\delta V}\int_{\delta V}\mathbf{u}(x,y,\eta,t)dxdydz \tag{1}$$

where $dV_s = dVf_s(h,h_x,h_y,h_t)dhdh_xdh_ydh_t$, $f_s = f_s(h,h_x,h_y,h_t)$ is a multiple probability distribution function, $\eta = z/h$, δV is an arbitrary volume enclosed in $dV = L_xL_ydz$ and containing the averaging volume dV_s as a whole, L_x, L_y are the characteristic scales of the flow in the parallel to the wall directions.

Statistical moment of an order m of the random function $\tilde{\mathbf{u}}(\eta,t,h,h_x,h_y,h_t)$ are determined as follows

$$\overline{\tilde{u}}^m(z,t)=\int\tilde{u}^m(\eta,t,h,h_x,h_y,h_t)f_s(h,h_x,h_y,h_t)dhdh_xdh_ydh_t \tag{2}$$

This theory has been developed for accelerated and decelerated turbulent boundary layers. The turbulence model has been testified in a case of zero, adverse and favourable pressure gradients.

The computed mean velocity profiles in adverse pressure gradient are shown in Figure 1 together with the experimental data by Nagano *et al* (1992). The input values and pressure gradient parameter, $p^+=\frac{\nu}{u_*^3}\frac{\partial p}{\partial x}$, are listed in Table 1.

To estimate the coherent structure geometry parameters the asymptotic model described the dynamic roughness surface has been derived from the Navier-Stokes equation as follows

$$\nu K_\alpha\left(\frac{\partial^2\alpha}{\partial x^2}+\frac{\partial^2\alpha}{\partial y^2}\right)+\nu K_{\alpha\alpha}\left(\alpha_x^2+\alpha_y^2\right)=-\frac{2u_*^2w_0^+a}{\nu\lambda^{+3}}\cos\alpha+K_\alpha\frac{\partial\alpha}{\partial t} \tag{3}$$

where $\alpha=\arctan(h_y/h_x)$ is the parameter characterised the dynamic roughness structure $K_\alpha=-w_0^+a\sin\alpha+\cos\alpha$, $K_{\alpha\alpha}=-w_0^+a\cos\alpha-\sin\alpha$. The turbulence theory constants have been calculated as $w_*^+=\kappa R_t^*\exp(-I_0)=0.14$, $\lambda_0^+=R_t^*/w_*^+=8.71$ for $\kappa=0.41$, where κ is the Karman constant, $R_t^*\approx1.22$ is the Reynolds number calculated with the dynamic roughness parameters.

[1] *Sochi Research Center for the Russian Academy of Sciences,*
8-A Theatralnya Str., Sochi, 354000, Russia,
Phone: +0078622975576, Fax: +0078622924411,E-mail: trounev@sochi.ru

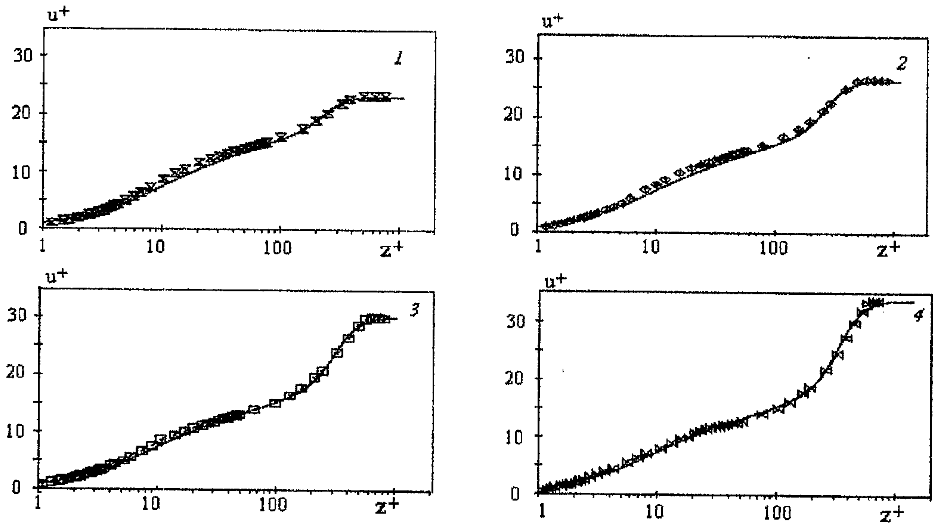


Figure 1: The mean velocity profiles in a turbulent boundary layer in adverse pressure gradients computed on the proposed model (the solid lines), and experimental data by Nagano *et al* (1992)

Table 1: The input parameters of the turbulent boundary layer [1]

NN	Viscosity, $\nu*10^5$, m^2/s	The friction velocity, u_τ, m/s	The boundary layer depth, H_{99}, mm	The pressure gradient parameter, p+
1	1.536	0.39	16.2	0.00898
2	1.576	0.307	24.6	0.0181
3	1.551	0.251	34.2	0.023
4	1.537	0.197	46.1	0.0251

Used the periodical solutions of the equation (3) one can calculated the characteristic scales of the coherent structures. The transversal length of coherent structures can be estimated as

$$\lambda_y^* = 2\pi / k_y = \sqrt{2}\pi\lambda_0^{+3/2}\nu / u_* \approx 114\nu / u_*$$

The predicted length scale is in a good agreement with the experimental value, $\lambda_y \approx 100\nu / u_*$, obtained by Kline *et al.* (1967). The transversal phase velocity of the dynamic roughness surface disturbances is $c_y = w_0^*$. The scale λ_x depends on the amplitude α_0 as

$$\lambda_x = \lambda_y^* \alpha_0 / \left| w_0^+ a \right| \approx 898\alpha_0 \nu / u_*.$$

For $\alpha_0 \approx 1$ the estimated streamwise length scale of coherent structures agrees with the experimental value, $\lambda_x \approx 1000\nu / u_*$, presented by Blackwelder & Eckelmann (1979).

The turbulent boundary layer model based on the coherent structure theory has been successfully used to estimate the flow parameters in the stratified surface layer with roughness density effect on the turbulent flow in case of 2D and 3D roughness elements [2].

References

[1] **Nagano, Y., Kasagi, N., Ota, T., Fujita, H., Yoshida, H. & Kumada, M.,** Data-Base on Turbulent Heat Transfer, *Department of Mechanical Engineering, Nagoya Institute of Technology, Nagoya, DATA No. FW BL004, 1992.*

[2] **Trunev, A. P.,** Theory of turbulence and model of turbulent transport in the atmospheric surface layer, *SRC RAS, 1999, 160 p.*

Coherent Vortical Structures in Limited Swirling Flows

V. N. Turick[1], R. A. Makarenko[1], A. V. Voskoboinick[2], V. A. Blohin[2]

The rational control of the liquids or gases mixing and energy dissipation processes in vortex chambers is impossible without detailed study of the differently scaling coherent vortical structures. These structures are able according to conditions to decrease the turbulence in swirling flows or to increase its influence on mass and heat transfer processes in cylinders of internal combustion engines, MHD generators, power-generating equipment, chemical technology etc.

The aerodynamic stand for experimental study of such currents contains vortex chamber including the tube with transparent wall which has internal diameter d_0=102 mm and length l_0=635 mm with fixed in various positions cylinder end. The air admission into internal tube cave is fulfilled through the single-jet changeable nozzles (flow cross section 41·25 mm^2) with different tangential and axial corners. Relative depth of the cylinder dead zone (between middle of the inlet nozzle and the end) is varied in range L/d_0=1.82…4.4. Reynolds number in conformity with mean admission velocity in the nozzle and its equivalent diameter is varied in range Re=40000…80000. Besides usual rate-of-flow and pressure meters aerodynamic stand is equipped the measuring complex which consists of devices for flow visualization, stroboscope and high-speed photographic instrumentation, pneumatic pickups, the Constant-temperature Anemometer (Fig.1).

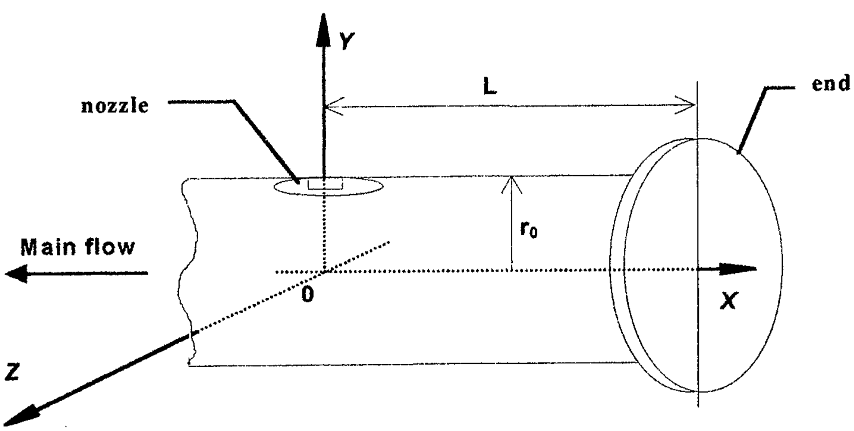


Fig.1 Vortex chamber scheme.

Experimental data show that there are several groups of steady coherent vortical structures in dead end part of vortex chamber: small-scale secondary currents like Taylor-Goertler vortices and "embedded" one into other large-scale vortex structures with different signs of average axial velocities along the dead end zone. On the typical diagram of axial velocity distribution in vertical plane with indication of relative velocity values, the white color corresponds to current in the direction of chamber dead end. Grey and black colors (the last one shows high speed) correspond to reverse current (Fig.2).

The mechanism of the large-scale structures forming is submitted. The higher velocity levels of the peripheral spiral flow in comparison with the second part of air current to the end allow to consider influence of the most energetic first flow on the dead zone current as determinant. Therefore in this work the data generalization, describing geometry of the main peripheral spiral flow and axial average velocities distribution in characteristic cross-sections of the chamber dead end zone, is offered. In addition, the empirical formulas for tangential velocity distribution along the dead end zone are received.

[1] *Kiev, Ukraine, National Technical University of Ukraine "KPI"*
[2] *Kiev, Ukraine, Institute of Hydromechanics National Academy of Sciences of Ukraine*

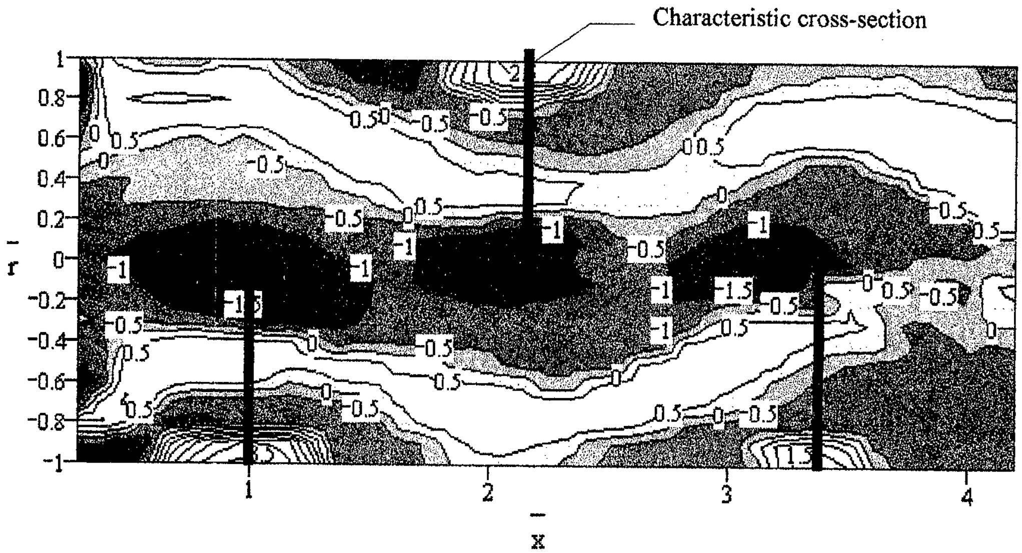


Fig.2. Axial velocity distribution along the dead end zone, Re=77600, L/d_0=4.4.
($\bar{x} = x/d_0$; $\bar{r} = r/r_0$; r – current radius).

Some results of measurement and visualization pictures for small-scale coherent vortices near the nozzle (admission zone of flows division), along the concave chamber wall and directly near the cylinder end (concentrated vorticity) are given.

This research has the purpose to accumulate geometric and cinematic characteristics of the dead end currents for assignment the conditions of unambiguity at the mathematical models of the chamber swirling flows.

A Deformed and Deformable Surface as a Means of Boundary Layer Characteristics Control

G.A.Voropaev[1], N.V.Rozumnyuk[2]

It is now generally recognized that coherent vortical exist in the turbulent boundary layer against a statistical background of velocity and pressure fluctuations.

The appearance of vortex structures of the scale of the boundary layer (Clain vortices, hairpin vortices) may by associated with the breakdown of the near-wall region (viscous sublayer) and their presence in different locations in the boundary layer is determined by their life-time, convective and diffusive transport downstream and through the boundary layer thickness.

Properties of these vortex structures depend significantly on the Reynolds number, as well as other factors, such as longitudinal pressure gradient, surface roughness, turbulence intensity in the flow.

Turbulent boundary layers over deforming surfaces have not been studied extensively to date. While there exist many experimental data concerning turbulent boundary layer structure over rough surfaces which with irnregular (sandy surface) or regular structure (longitudinal (riblets) or transverse(waves, trenches)), these data have not been generalized from the position of existence of the vortex structures in the flow.

Unlike boundary layers without a pressure gradient, coherent structures appear in the boundary layer over a wavy surface, and their scales correlate with scales of deformation of the surface.

Direct numerical simulations of viscous flows over a wavy surface have revealed conditions for formation of stable coherent vortices. Their size and vorticity magnitude depend on the Reynolds number, and width to depth ratio, as well.

Over an oscillating surface, disturbances are generated in dependence on the amplitude and frequency, as well as phase shift between normal and longitudinal amplitude of the surface. Over a deformed surface, there are not only changes in the level of intensity of turbulent fluctuations, but also qualitative changes in components of the turbulent energy balance equation. In dependence on phase shift and frequency, both sign of turbulent diffusion sing of viscous diffusion may change.

At the same time, it has been shown experimentally, if a standing wave is generated on the streamlined surface, so that there is no longitudinal displacement, than no changes occur in spectral density of pressure fluctuations, in the considered ranges of amplitudes, wave length and frequency. That indirectly makes questionable the hypothesis about the relation between low-frequency part of the spectrum with the frequency of breakdown of viscous sublayer.

[1,2] *Institute of Hydromechanics UNAS, Zhelyabov str., 8/4. 03057, Kiev, Ukraine.*

E-mail: vga@tbl.kiev.ua

Wall Pressure Fluctuations on the Cylinder Streamlined under an Angle of Attack

V. A. Voskoboinick[1], V. T. Grinchenko[1], A. P. Makarenkov[1]

At operation of flexible extended towed arrays there are regimes, at which they are streamlined by a flow of a liquid under corners of attack up to 10 - 15 degrees. The development of a turbulent boundary layer on them has the certain differences from axial symmetric flow. Also is devoted to definition of the characteristics of wall pressure fluctuations on the flexible extended cylinder, streamlined under a corner of attack, offered experimental work.

The researches were made on range of Institute of the Hydromechanics NAS of Ukraine in the channel of length about 4000 m, width about 40 - 50 m and depth about 5 m, method of towage of the flexible extended cylinder, lengthening L/d = 690, on depth 1.2 m on an original methods in detail stated in the report. The researches were carried out at speed of towage 3.0 m/s and corners of attack $\alpha = 0°$; $2°$; $4°$; $8°$. A possibility of turn of measuring section about the axis with a step in $90°$, for study of development of a boundary layer on an azimuth of the cylinder also was stipulated.

Air-filled polychlorovinil cover, diameter d = 2a = 29 mm was used as measuring section the environment. Membrane-type piesoceramic transducers of pressure fluctuations flush with a wall, diameter of a sensitive surface of 1.6 mm (d = d*u/n = (174 - 206), were established on length of measuring section. In measurements 7 single transducers and correlation block from 6 transducers were used. In the correlation block 4 transducers of pressure fluctuations were established on forming of the cylinder and two transducers were in a diametrical plane of the fourth transducer on a circle of the cylinder under corners $47°$ and $51°$ relatively the central transducer.

The experimental researches were made at $Re_x = xU_\infty / \nu = (1-6) \cdot 10^7$; x/a = (1048 - 1344); $\delta / a = (2.94 - 3.06)$; $\delta^* = (6.57 - 6.83) \cdot 10^{-3}$ m; $u_\tau = (0.11 - 0.13)$ m/s, for $\alpha = 0°$.

Measurements and the analysis of experimental results was made with the help of a complex of the equipment of firm Bruel and Kjaer (Denmark) and on universal computer facilities, under the appropriate programs and methods.

Character and the meanings of the power spectral density of pressure fluctuations remained practically constant, irrespective of positions of transducers on length of the cylinder, for the given corner of attack. The power spectra of the wall pressure fluctuations on low frequencies with increase of a corner of attack grows and does not change in the field of high frequencies. With increase α the boundary layer is sated by large-scale low-frequency vortical structures, which have the large energy.

Correlations of vortical structures on forming of the cylinder with increase of a corner of attack of a flow essentially falls, is especial for frequencies $\omega^* = \omega d / U_\parallel < 8$ ($U_\parallel = U_\infty \cos\alpha$). The increase of a corner of attack results in decrease of coherence, similarly to increase of separation between transducers, leaving constant frequency $\omega d / U_\parallel \approx 1.5$, where the maximum of the coherence is observed. As against increase of separations between transducers, the increase of a corner of attack, in a researched range, does not result in narrowing frequency region, in which is observed the coherence. Thus, the increase of a corner of attack of a flow of the flexible cylinder an insignificant influences on the correlation of the high-frequency small-scale pressure-produced vortical structures and reduces it at large-scale high-speed systems. Thus, most wall pressure fluctuations on frequency $\omega^* = 1.5$.

The power spectrum of wall pressure fluctuations of the turbulent boundary layer formed at axial symmetric flow of the flexible extended cylinder, remains constant on a circle of this cylinder. At a deviation from axial flow, the power spectrum undergoes changes. In a frontal part, on an azimuth of the cylinder, where $0° < \varphi < 90°$ and $270° < \varphi < 360°$, correlation vortical structures, without dependence from a corner of attack, there are low-frequency systems causing the power spectrum, at removal from a forward critical point ($\varphi = 0°$ or $\varphi = 360°$), decreases. When the corner φ reaches

[1] ***Kyiv, Ukraine, Institute of Hydromechanics National Akademy of Sciences of Ukraine***

90° and by virtue of symmetry 270°, the spectrum has the minimal meaning. At transition in a stern part on azimuth of the cylinder, where $90^{\circ} < \varphi < 270^{\circ}$, the power spectrum begins to grow and reaches maximal meaning at $\varphi = 180°$. This implies, that the power spectrum of pressure fluctuations of the turbulent boundary layer formed on the flexible cylinder, streamlined under corners of attack, remaining constant in the field of high frequencies $\omega^{*} = \omega d / U_{\|} > 6$, has in low-frequency area minimal meaning(importance) at $\varphi = 90°$ $(270°)$ and maximal meaning in a stern part on circle of the cylinder at $\varphi = 180°$.

The similar tendencies of change of the power spectra of wall pressure fluctuations and functions of coherence in the field of low frequencies and invariance them in high-frequency area were observed for all researched not zero corners of attack. But with increase it is shown more clearly.

Conclusions:

1. The program and original technique is developed, the measuring breadboard model is created and the experimental researches on study of development of a turbulent boundary layer on the flexible extended cylinder, streamline under a corner of attack are carried out(spent).
2. Is established, that character and the meanings of spectral power of wall pressure fluctuations, for the given corner of attack, do not depend on a position of transducers on length of the cylinder of the given lengthening.
3. It is revealed, that the power spectrum of pressure fluctuations with increase of a corner of attack grows in the field of low frequencies and varies unsignificantly in a high-frequency range.
4. At axial symmetric flow of the cylinder the power spectrum does not undergo changes on an azimuth of the researched cylinder.
5. The power spectrum of pressure fluctuations on the flexible extended cylinder, streamlined under a corner of attack, remaining practically constant in the area of high frequencies, in low-frequency area has the minimal meaning on a surface, where $\varphi = 90°$ or $270°$ and maximal - in a stern part of the cylinder, where $\varphi = 180°$, thus with growth α this difference is shown in the greater degree.
6. The correlations of vortical structures on forming of the cylinder, with increase of a corner of attack of a flow, decreases, mainly, large-scale.
7. Most correlation vortical systems are the low-frequency large-scale eddies producing pressure fluctuations on a surface of the cylinder on frequency $\omega^{*} = 1.5$, without dependence from a corner of attack.
8. The turbulent boundary layer of the flexible extended cylinder, streamlined under a corner of attack, with increase of a corner of attack becomes more sated with large-scale vortical structures having higher velocity of convection.
9. The presence on a circle of the cylinder of large-scale vortical systems which are producing in the basic contribution to low-frequency fluctuations, reaches a maximum in a fodder part of the cylinder and minimum for $\varphi = 90°$ and $270°$, at angular flows of the cylinder.

Utilization of Streamwise Vortices Intrinsic to Flow Evolution for Boundary Layer Control

Nina F. Yurchenko*

Background

Typical kinds of fluid motion can often be found, predicted and associated with certain flow conditions. For instance, such is the Karman's vortex street in flows past blunt bodies or streamwise vortical structure developing in near-wall flows affected by body forces. In particular, the second type of the vortex motion appears to represent a family of flows connected with curvature and rotation like boundary-layer flows over concave surfaces or rotating plane Couette flows, see e.g. Saric (1994), Yurchenko & Delfs (1999), Andersson (2000).
Nowadays, a role of streamwise vortices in fluid transport near a wall is recognized; their influence on heat transfer and flow separation was analysed using the Goertler instability approach, Yurchenko (1998); Yurchenko, Rivir (2000). It defines further exploration of the problem in the frame of practicability to use the self-organized and dominating vortical structure.
Thus, the objective of the present work is to get an insight into natural and forced evolution of streamwise vortices aimed at an optimal surface-flow interaction from a viewpoint of energy outlay as well as convenience and flexibility of the flow management. In other words, it means the development of a gentle way to controllably vary fluid motion space-time scales without drastic changes of the vortex dynamics.

Results and Discussion

Experimental and numerical studies of a boundary-layer flow over a concave surface were carried out on a basis of the classical Goertler stability theory (Saric, 1994) and receptivity approach (Yurchenko & Delfs, 1999). Streamwise vortices were generated with a controlled scale and intensity due to an imposed boundary condition in a form of a surface temperature gradient ΔT periodic in a spanwise direction. There were considered three cases of the vortex structure development:

Case 0, a reference case of Goertler vortices naturally evolving in a boundary layer during a laminar-turbulent transition;

Case 1, second mode excited with a non-dimensional vortex scale Λ=84 linearly amplified according to the Goertler stability diagram, and harmonics thereof;

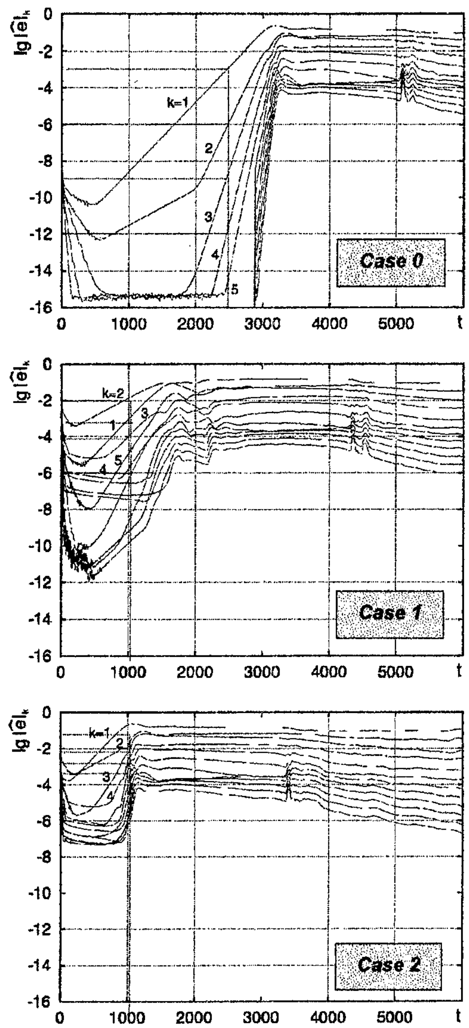
Case 2, slightly irregular vortical structure generated that implies additional stimulation of a spectrum including the most amplified first mode, Λ=236;

In matched experiments, the mentioned boundary condition was realized using electrically heated longitudinal wires flush-mounted in a test plate. Varying a distance between the neighbouring heated wire-strips and a value of applied voltage, one can change a space scale and intensity of the induced vortical system. In practice, *Case 2* was realized due to one of the heated strips insignificantly shifted along *z* thus modelling imperfectness of the experimental arrangement.

Earlier reported flow topology and scale transformation during successive phases of the vortical structure development is supplemented here (see Figure) with evolution of amplitudes of separate

* Dep. of Thermal and Fluid Dynamic Modeling
Institute of Hydromechanics
National Academy of Sciences
8/4 Zheliabov St., 03057 Kiev, Ukraine

E-mail: nina.yurchenko@mbox.com.ua



modes of various flow variables. Here, mode-amplitude of the k^{th} mode, $|\hat{e}|_k = ({}_0\int^{\infty}|\hat{e}_k(y)|^2\, dy)^{1/2}$. Thus, $|\hat{e}|^2_k$ is a measure of y-averaged energy, contained in each of the modes of the signal e^2 such that the sum $\sum |\hat{e}|^2_k$ over all $k \neq 0$ represents the y-averaged mean square value of the flow energy.

The shown mode-diagrams display qualitatively similar evolution of the vortical system that corresponds both calculated and measured velocity fields in considered cases. However the "growth phase" (see Yurchenko Delfs, 1999), i.e. a developed stable vortical system can be initiated and maintained longer due to the thermal control from the surface than in the reference case. It was estimated that compared to the reference Case 0 (natural boundary layer development), the downstream distance of a developed and stably sustained vortical system extends by about 39% for the second harmonic forcing (Case 1) and by about 22% for the "irregular excitation" (Case 2).

Conclusions

Inherently developing streamwise vortices are easily controllable with accounting for the correlation between the excitation mode and basic flow parameters. It relates to the scale, intensity and the growth rate of vortical structure. Only a slightly different arrangement of the controlling heating elements causes significant changes in the flow even in the late stages of the vortex evolution.

Acknowledgements

The author acknowledges with thankfulness the support by the European Office of Aerospace Research and Development, AFOSR, AFRL under the Contract F61775-99-WE075, the stimulating interest to the work of Dr. C. Raffoul as well as contributions by Dr. J. Delfs, DLR, Germany, in part of numerical analysis, and Mr. Lian Yuan, Tianjin University, in experimental investigations.

References

Saric, W. S. Goertler vortices (1994) *Annual Review of Fluid Mechanics*, **26**, pp 379-409.

Yurchenko, N., Delfs J. (1999) Optimal control of boundary layers under body forces, *Proc. IUTAM Symposium on Laminar-Turbulent Transition, Sedona, U.S.A., September 13-19.*

Andersson, H.I. (2000) Organized Vortical Structures Induced by System Rotation: Computer Experiments and Mathematical Modelling. *In: Organized Vortical Motion as a Basis for Boundary Layer Control, Kiev, September 20-22.*

Yurchenko, N., Rivir, R. Improvement of the turbine blade performance based on the flow instability and receptivity analysis, 2000, *8th Int. Symp. on Transport Phenomena and Dynamics of Rotating Machinery (ISROMAC-8), Honolulu, Hawaii, March 26-30, 2000.*

Yurchenko, N.F. Optimization of heat transfer control based on a receptivity approach, 1998, *Proc. Turbulent Heat Transfer Conference, Manchester, UK*, P72-P81.

Surface Frictional Processes and Non-Local Heat Transfer in Shear-Free Convective Boundary Layers

S. Zilitinkevich[1,2], A. Grachev[2], and J.C.R. Hunt[3]

During the last several decades the surface frictional processes in the shear-free convective boundary layer (CBL) were considered conceptually in the spirit of the Prandtl (1932) theory of free convection, implying the ideas of universal chaotic turbulence and local correspondence between turbulent fluxes and mean gradients. Accordingly the fluxes of heat and water vapour in the atmospheric surface layer were parameterised disregarding gross features of the CBL. Conventional practical tools were either the surface-layer similarity theory or simple downgradient turbulence closure models. However, in strong convection regimes the concepts of completely chaotic turbulence and local transport underlying conventional theories break down. At very high Rayleigh numbers buoyancy-driven large-scale semi-organised structures develop and embrace the entire CBL. In the atmosphere, they consist of comparatively narrow strong uprising plumes surrounded by wider and weaker downdraughts. The surface layer includes the CBL-scale convergence (towards plume axis) flows and plays the role of a feeder layer for plumes. The surface-layer flow patterns can be treated as internal boundary layers of radial geometry strongly affected by the buoyancy forces. Generally convergence flows superimpose on (and interact with) the mean wind. In the shear-free regime they yield their own velocity shears whose absolute values are characterised by the "minimum friction velocity", U_*. The shear-free convective turbulence turns out to be dependent on the surface friction, which can be characterised by the "minimum Monin-Obukhov length", $L = L_* \equiv U_*^3 / B_s$, where B_s is the buoyancy flux at the surface (in contrast to the classical formulation that implies $L \to 0$). Then turbulent mixing and heat/mass becomes strongly affected by gross features of the CBL, such as the CBL depth, h, and the surface roughness length, z_{0u}. Theoretical, numerical and experimental analysis of the above mechanism have been made by Businger (1973a,b), Schumann (1988), Sykes et al. (1993), Siggia (1994), Zilitinkevich (1995, 1997), Grachev et al., (1997), Zilitinkevich et al. (1998), Akylas et al. (2000). In the present paper the problem in question is comprehensively discussed. A reasonably simple model of the non-local friction and transport processes in strong convection regimes over rough surfaces is presented and validated empirically for $z_{0u} / h < 10^{-4}$ using experimental and large-eddy-simulation data on the minimum friction velocity and the heat/mass transfer. The model results in an improved parameterisation of surface fluxes in atmospheric problems.

[1]*Dept. of Earth Sciences, Uppsala University, SE-752 36, Uppsala, Sweden*
[2]*A.M. Obukhov Institute of Atmospheric Physics, RAS, 109017 Moscow, Russia*
[3]*Dept. of Space and Climate Physics, University College London, WC1H OAH, UK*