JR SMIT 2015

Japan-Russia Workshop

SUPERCOMPUTER MODELING, INSTABILITY AND TURBULENCE IN FLUID DYNAMICS

March 3-6, 2015

Keldysh Institute for Applied Mathematics
Russian Academy of Sciences, Moscow, Russia

BOOK OF ABSTRACTS

Moscow-2015

Organization

Keldysh Institute for Applied Mathematics

Russian Academy of Sciences (KIAM RAS)

Workshop coordinators

Academ. B. Chetverushkin, KIAM RAS, Moscow, Russia

Prof. K. Ishii, Nagoya University, Nagoya, Japan

Prof. I. Menshov, KIAM RAS, Moscow, Russia

Supported by

Russian Foundation of Basic Research (RFBR)

Japan Society for the Promotion of Sciences (JSPS)

JR SMIT2015 Workshop - Agenda			
	04/03/2015 Wednesday	***05/03/2015 Thursday***	***06/03/2015 Friday***
	Session I Chairs: K. Ishii, I. Menshov	**Session IV** Chairs: T. Ishihara, V. Gushchin	**Session VII** Chairs: K. Izawa, A. Lipanov
10:00 - 10:30	B. Chetverushkin	K. Ishii	A. Lutsky
10:30 - 11:00	V. Chechetkin	A. Lipanov	Y. Fukunishi
11:00 - 11:30	Y. Nagata	V. Podryga	S. Frolov
11:30 - 12:00	D. Puzyrkov	V.Levchenko	A. Lugovsky
12:00 - 13:30	**Lunch**	**Lunch**	**Lunch**
	Session II Chairs: S.Adachi, T.Kozubskaya	**Session V** Chairs: K.Kitamura, T.Elizarova	**Session VIII** Chairs: Y.Fukunishi, I.Semenov
13:30 - 14:00	H. Suito	S.Izawa	T.Kozubskaya
14:00 - 14:30	V. Gushchin	I.Semenov	T.Ishihara
14:30 - 15:00	K. Kitamura	S.Budaev	A.Babakov
15:00 - 15:30	T. Elizarova	A.Severin	A.Gorobets
15:30 - 16:00	**Coffee break**	**Coffee break**	**Coffee break**
	Session III Chairs: T.Miyazaki, A.Babakov	**Session VI** Chairs: H.Suito, A.Lutsky	**Session IX** Chairs: S.Taguchi, V.Chechetkin
16:00 - 16:30	P. Matyushin	S. Adachi	T.Miyazaki
16:30 - 17:00	D. Saburin	I. Kudryashov	T.Kudryashova
17:00 - 17:30	S. Taguchi		I.Menshov
17:30 ~	**Reception**		**Banquet**

Table of contents

NUMERICAL STUDY OF THERMOACOUSTIC OSCILLATIONS IN A CLOSED TUBE BY TRACING FLUID PARTICLES

S. Adachi[1], K. Ishii[2]
[1]Tokyo International University, Kawagoe, sadachi@tiu.ac.jp
[2]Nagoya University, Nagoya, ishii@cc.nagoya-u.ac.jp

Spontaneous thermoacoustic oscillations of a gas in a tube are observed when there is a large temperature gradient along the tube axis. Energy conversion of the oscillations of a helium gas in a closed tube is numerically studied by tracing fluid particles. This paper reveals the regions where fluid particles serve as prime movers or heat pumps.

The wall temperature near both ends of the tube is room temperature 300K and that of the central region is 20K. The ratio of the length of the hot part to that of the cold part is ξ. We did numerical simulations for various values of ξ.

Since the ratio of the radius to the length of the tube is very small $r_0 /L= 2.7\times10^{-3}$, we assume that the flows are axisymmetric. The basic equations are the axisymmetric compressible Navier-Stokes equations of a perfect gas. The block pentadiagonal matrix scheme[1] is employed to solve the basic equations. The second-order accurate three-point backward implicit scheme is used for the time development. Convective terms are evaluated by using fourth-order central differencing and viscous terms by second-order central differencing. We use a rectangular grid system consisting of 300 points along the tube length and 36 points on the tube radius. The boundary conditions on the tube wall are non-slip and isothermal, and no pressure gradient in the normal direction of the wall is applied.

We identify three oscillation modes of steady oscillations observed at ξ between 0.26 and 4.4.: a symmetric (second) mode with pressures at both tube ends oscillating in phase, a shock-wave mode and an antisymmetric (fundamental) mode with pressures at both ends oscillating pi out of phase [2].

Temporal evolution of thermodynamic quantities of fluid particles is obtained from the flow field. The amount of work done by each fluid particle during one period is estimated. In the symmetric mode (ξ = 0.4) with smaller pressure amplitudes, fluid particles move in the vicinity of their starting points. Fluid particles in the region of finite temperature gradient serve as prime movers, and those near the end walls serve as heat pumps. Fluid particles do almost no work in the center region. In the antisymmetric mode (ξ = 1.0), displacement of fluid particles during one period is large. Both in the region of finite temperature gradient and in the tube center, fluid particles near the wall serve as heat pumps and those near the axis serve as prime movers. Fluid particles near the tube end wall serve as heat pumps.

References

1. Shida Y, Kuwahara K, Ono K and Takami H. AIAA J. 25. 408-413, 1987.
2. Ishii K, Kitagawa S, Ishigaki M and Adachi S, Fluid Dyn. Res. 46, 061408(pp.14), doi:10.1088/0169-5983/46/6/061408, 2014.

NUMERICAL SIMULATION OF FLOW SEPARATION IN NOZZLES.

A.V.Babakov

Institute for Computer Aided Design, RAS
2-nd Brestskaya st., 19/18, Moscow 123056, Russia, babakov@icad.org.ru

The investigation of flow in nozzles is the important for prediction of the negative phenomena such as the unsteady flow separation and intensive pressure pulsations. The numerical simulation of similar flows is based on a conservative finite-difference flux method for unsteady model of viscous heat-conducting compressible gas. The behavior of flow and it stability are depend on the relationship between the total pressure in convergent part of nozzle and the ambient pressure.
The numerical investigations are carried out for supersonic plane and conical nozzles. For plane nozzle the field of Mach number local value is shown on fig.1 as example for flow without separation and with boundary-layer separation.

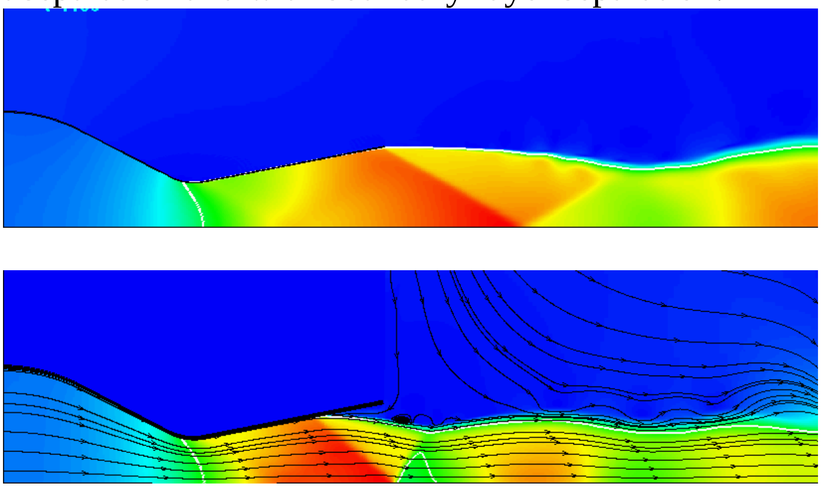


Fig.1.

For supersonic conical nozzle the numerical simulation is realized in three-dimension formulation. For examples, the field of density gradient and instantaneous streamlines are represented on fig.2 for over-expanded regimes, which is characterized the deep penetration of unsteady flow separation into nozzle with shock-wave systems.

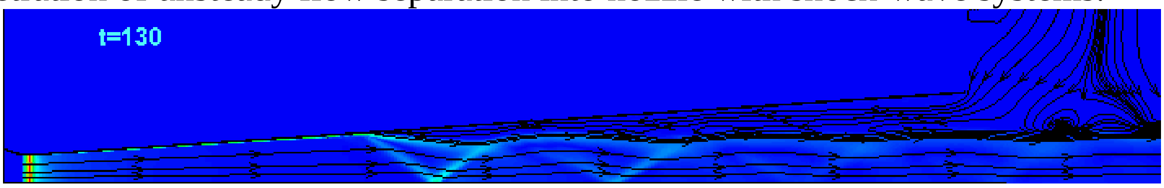


Fig.1.

The computations are performed using parallel algorithms implemented on a cluster-architecture supercomputer.

The investigations were carried out at RFBR financial support (grant 15-51-50023 ЯФ).
The calculations were realized on JSCC RAS supercomputers.

TURBULENT BOUNDARY LAYER CONTROL BY FRACTAL SURFACE

V. P. Budaev[1], I. S. Menshov[2], M. A. Brutian [3], A. V. Volkov [3], A. M. Zhitlukhin[4], I. Yu. Kudriashov[2], A.V. Severin[2], A.V. Karpov[1], N. S. Klimov[4], V. L. Podkovyrov[4]

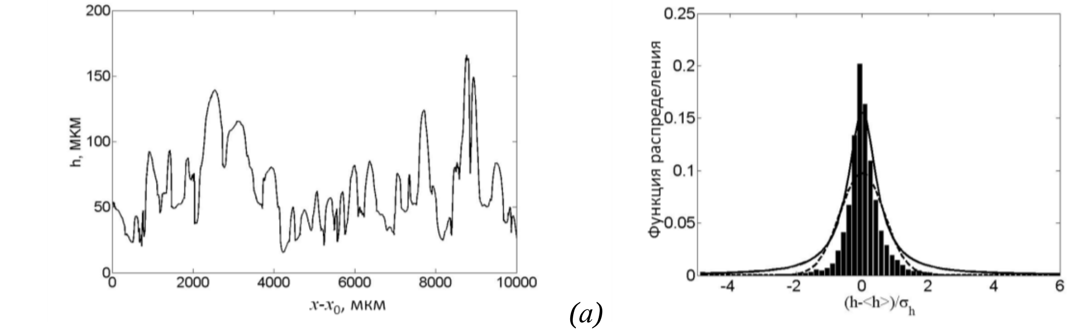
[1] NRC «Kurchatov Institute», Moscow, Russia, budaev@mail.ru

[2] Keldysh Institute of Applied Mathematics RAS, Moscow, Russia, menshov@kiam.ru

[3] TsAGI , Zhukovsky, Moscow Region, Russia

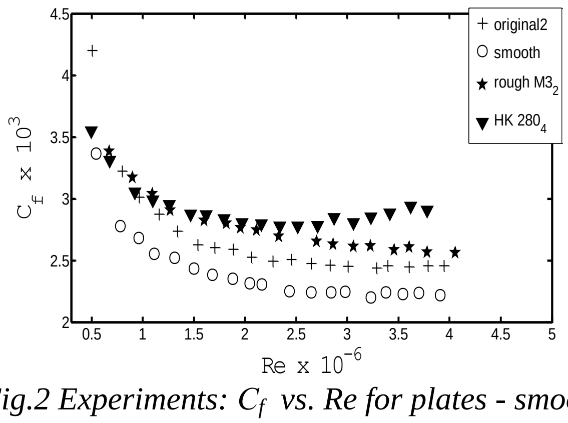
[4] SRC RF TRINITI, Troitsk, Moscow, Russia

Turbulent boundary layer over the fractal surfaces of the specific granularity and long-range correlation is studied in the T-36I wind tunnel [1]. The fractal surfaces of roughness from ~0.5 to ~200 micrometers is formed in the process of the plasma-surface interaction in fusion device QSPA-T. The specific property of fractal surface is non-Gaussian statistics of relief heights distribution (Fig.1). Within broad range of Reynolds numbers it was observed (Fig.2) the drag reduction c_f over the fractal plate comparably with the c_f for the abrasive surface (trivial roughness with the Gaussian statistics of heights) of the same roughness. For the fractal surface the scaling index ν of the drag coefficient $c_f \sim Re^{-\nu}$ is close to the ν for the smooth plate.



(a) *(b)*

Fig.1.(a) The heights profile of the fractal surface. (b) Distribution function of the profile heights. The Gaussian (dotted line) and the Cauchy (solid line) distribution are shown for the comparison



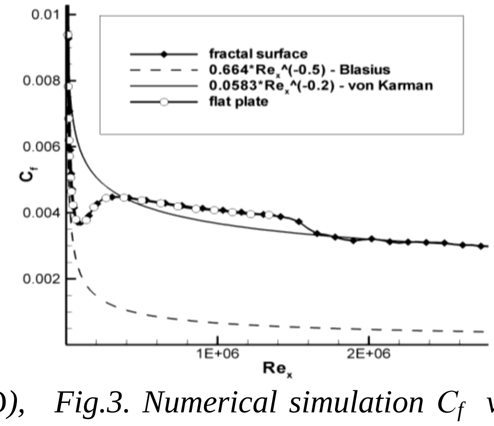


Fig.2 Experiments: C_f vs. Re for plates - smooth (O), virgin (+), fractal (stars), abrasive (triangels)

Fig.3. Numerical simulation C_f vs. Re for fractal and smooth plates

Experimental observations are in agreement with the numerical simulation of the TBL over the fractal surface, Fig.3. Numerical simulations based on the 3D Reynolds equations and the turbulence model of Spalart-Almaras solved by the finite volume method have shown a reduction of the c_x over the fractal surface compared with no reduction for the abrasive surface (the Gaussian statistics of heights).

1. M. Brutyan, V. Budaev, A.Volkov, I. Menshov e a., TsAGI Science Journal, 2013, 4 , 16.

Regularized shallow water equations in numerical modeling of tank sloshing and tsunami propagation

Bulatov Oleg[1], **Saburin Dmitrii**[1], **Elizarova Tatiana** [2]

[1]Moscow State University, Moscow, Russia
[2]Keldysh Institute for Applied Mathematics Russian Academy of Sciences, Moscow, Russia

The present paper briefly describes the new numerical method for shallow water (SW) flow simulations and related numerical results. The numerical method is based on a specific form of averaging, or regularization, of SW equations. This approach is closely connected with quasi gas dynamic algorithms developed earlier. The numerical algorithm for regularized SW system implements an explicit in time finite volume form, that is efficient for non-stationary flow modeling. The method is easily adopted for parallel computations and is naturally generalized for unstructured meshes [1].

We consider the problem of the sloshing in ice-breaker's tanks after impact interaction with ice barrier. The calculations were made for 10-percent tanker fill with bottom shape. We investigate two variants of vessel velocity: directed along the plane of symmetry of the tank and with angle to it. The obtained non-stationary pictures of the free surface motion, stream lines and pressure distribution on the tank walls are analyzed and compared with known numerical simulations using Navies-Stokes (NS) equations [2]. Our results correspond well with NS simulations, but are sufficiently cheaper compared with full flow modeling, including the free-surface calculations and turbulent phenomena models.

As a next example the numerical results for a problem of 'tsunami runup onto a complex beach is shown. Problem statement and the etalon results corresponds with the experiment, made in Research Institute for Electric Power Industry in Abiko, Japan. This experiment models the Okushiri tsunami of 1993 in the Monai Valley. Numerical results shows the nice possibilities of the present SW calculations for the practical applications in runup problems [3].

This work was supported by grants RFFI 13-01-00703a and 15-51-50023 Φ.

[1] T. G. Elizarova, O. V. Bulatov. Computers and Fluids 2011, N 46, pp.206-211
[2] T. G. Elizarova, D. S. Saburin. Scientific visualization, Vol. 4, pp.118-135, 2013.
[3] O.V. Bulatov. PhD thesis, Moscow, Keldysh Institute of Applied Mathematics. Dec.2014

MODELING OF THE EARTH'S ATMOSPHERE

V. Chechetkin and I. Mingalev
KIAM RAS, Moscow, chechetv@gmail.com

The regional non-hydrostatic mathematical model of the neutral wind system of the lower atmosphere is utilized to investigate the initial stage of the origin of cyclonic vortices at tropical latitudes of the northern hemisphere. The model produces three-dimensional distributions of the atmospheric parameters in the height range from 0 to 15 km over a limited region of the Earth's surface. The dimensions of the simulation domain in longitudinal and latitudinal directions are 32° and 25°, respectively. The model takes into account heating / cooling of the air due to absorption / emission of infrared radiation, as well as due to phase transitions of water vapor to micro drops of water and ice particles, which play an important role in energetic balance. The finite-difference method and explicit scheme are applied for solving the system of governing equations. The calculated parameters are determined on a uniform grid. The latitude step and longitude step are equal to 0.08°, and height step is equal to 200 m. Time-dependent modeling are performed for the cases when, at the initial moment, the simulation domain is intersected by the intratropical convergence zones with different configurations. Calculations are made for various cases in which the initial forms of the intratropical convergence zone are different and contained convexities with distinct shapes, which are consistent with the results of satellite microwave monitoring of the Earth's atmosphere.

The results of modeling indicate that the origin of a convexity of the form of the intratropical convergence zone, having the dimension of 800-1000 km, can lead to the formation of distinct large-scale vortices, in particular, a cyclone, cyclone-anticyclone pair, and pair of cyclonic vortices, during the period not longer than three days. The radii of these large-scale vortices are about 400-600 km. The horizontal wind velocity in these vortices achieves values of 15-20 m/s.

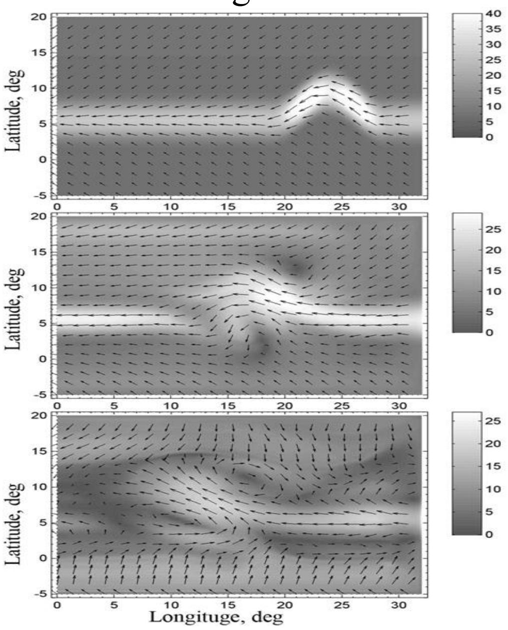


Fig. The distributions of horizontal component of the air velocity (m/s) at the altitude of 600 m, assigned at the initial moment (top panel), computed 12 hours after the beginning of calculations (middle panel), and computed 27 hours after the beginning of calculations (bottom panel). The results are obtained for the fourth initial configuration of the intratropical convergence zone.

Numerical simulation of subsonic turbulent viscous compressible flows using quasi-gas dynamic equations

Shirokov Ivan[1], **Elizarova Tatiana** [2]

[1]Moscow State University, Moscow, Russia
[2]Keldysh Institute for Applied Mathematics Russian Academy of Sciences, Moscow, Russia

We report our results of numerical modeling of laminar and turbulent subsonic viscous compressible gas flow based on quasi-gas dynamic (QGD) equations.The quasi-gas dynamic (QGD) system can be interpreted as the Navier-Stokes equation system averaged or smoothed, over some small time or space interval. The smoothing give rise to strongly non-linear additional dissipation terms proportional to a small parameter τ.

The τ-terms reproduce a new kind of subgrid-type dissipation. Similar to sub-grid dissipation in LES models, they smooth, or average the fluctuations of flow parameters on a time-space scale depending on discretization. The sub-grid dissipation in QGD equations differs from the turbulent Smagorinsky viscosity, as the τ-terms have different mathematical structure and properties. Additional terms appear not only in the momentum and energy equations, but also in the continuity equation that models the turbulent mass-diffusion, which is inherent to turbulent mixing. In the boundary layer the τ-terms vanish.

As an example of a laminar-turbulent transition in a free-flow we examined the Taylor-Green vortex decay. Here numerical results are compared with DNS and LES reference data for laminar flows with Reynolds numbers Re=100 and 280 and for the turbulent vortex decay regimes for Re=1600 and 5000 with Mach number $Ma = 0.1$ [1].

Couette flow simulations are performed for Reynolds numbers $Re = 300$, 3000 and 4200 and Mach number $Ma = 0.5$. By disturbing the initial laminar velocity profile we obtain the nonstationary turbulent flow pattern for $Re = 3000$ and 4200, and stationary laminar flow for $Re = 300$. The calculations were carried out on a highly-parallel computer.

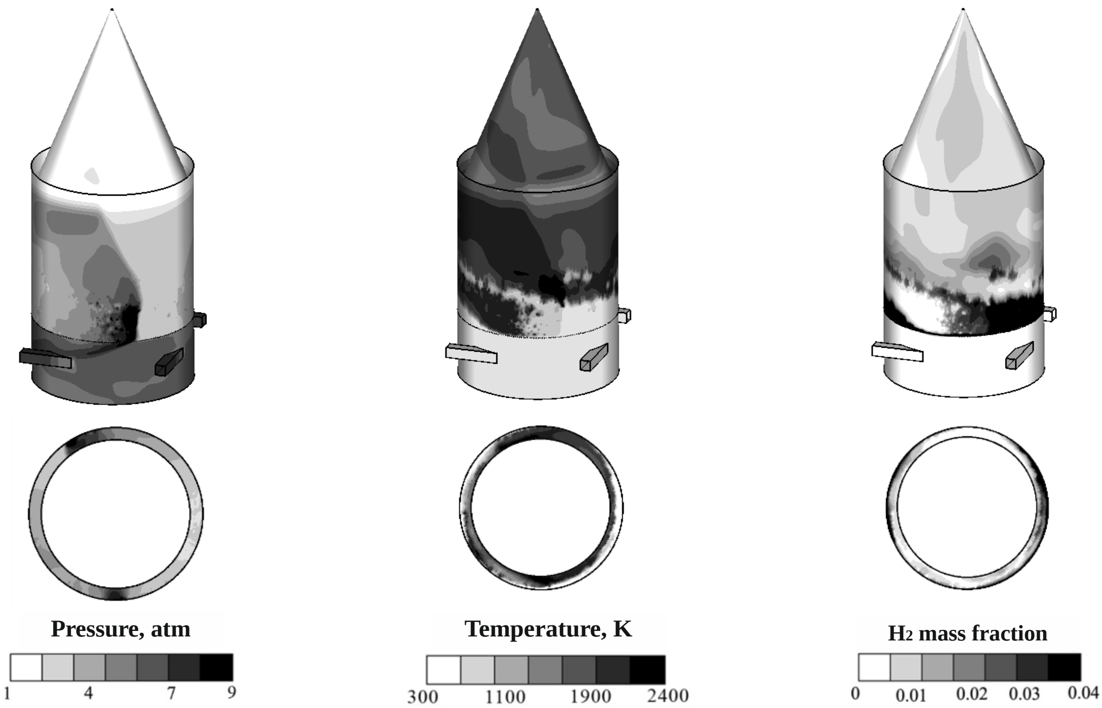
This work is supported by grants RFFI 13-01-00703a and 15-51-50023 Φ.

[1] I.A. Shirokov, T.G. Elizarova. *Journal of Turbulence* Vol.15, N 10: 707–730, 2014.

COMPUTATIONAL STUDIES OF ROTATING DETONATION ENGINE

S. Frolov, A. Dubrovskii, V. Ivanov
[1]*Semenov Institute of Chemical Physics, Moscow, smfrol@chph.ras.ru*

Presented in the paper are the results of 3D numerical simulations of the operation process and thrust performance of air-breathing hydrogen-fueled rotating detonation engine with separate delivery of fuel and oxidizer. The simulation is based on the coupled RANS – Monte-Carlo approach taking into account finite-rate turbulent and molecular mixing with multistage chemistry, turbulence – chemistry interaction and wall effects. The thrust is determined as an integral of pressure force and viscous stress over all solid walls of the engine. The calculations are aimed at the design optimization of an experimental engine prototype operating at mass flow rates of fuel components up to 10 kg/s. The calculation campaign includes variation of the design of air flowpath and hydrogen injectors, combustor dimensions, nozzle shape, etc. The underlying physical and mathematical model has been thoroughly validated against experimental data for the engine prototype of basic configuration with the annual combustor of 406 mm in diameter and a gap of 25 mm width (see figure below). The operation processes with one to multiple detonation waves simultaneously rotating in the annular combustor have been obtained. The predicted fuel-based specific impulse for the engine of basic configuration (~3000 s) agreed well with the measured value. As a result of design optimization the predicted specific impulse attained 4200 s.



NUMERICAL STUDY ON ONSET OF TURBULENCE USING INTERACTION BETWEEN STREAKY STRUCTURE AND JET

Y. Fukunishi[1], Y. Nishio, S. Izawa, J. Yoshikawa
[1]Tohoku Univ., Sendai, fushi@fluid.mech.tohoku.ac.jp

Numerical simulation is carried out to investigate the destabilizing process of a boundary layer. The boundary layer is destabilized by making streaky structures in a flat plate laminar boundary layer to interact with a short-duration jet ejection from the wall.

It is well known that turbulent spots can be artificially generated in a laminar boundary layer by electric sparks[1)] or short-duration jet ejection[2)] from the wall. However, quite a strong stimulation to the flow is needed to trigger a transition which leads to the formation of a turbulent spot. In case of the jet ejection from the wall, the jet velocity must be very high, around the freestream velocity or more, in order to generate a spot. On the other hand, in a natural bypass transition, such strong jets are not there to trigger the transition. Instead, there exists a streaky structure in the boundary layer.[3)] We believe, a turbulent spot could be generated using a much weaker jet only if we knew the right conditions of the flow field against which we eject the jet.

In our numerical simulation, two flow fields are computed. The first one is to introduce a streaky structure into the laminar boundary layer, which starts from a Blasius boundary layer but has equally spaced small bumps set on the flat plate aligned in the spanwise direction. The second flow field starts from a boundary layer with streaky structures. A short-duration jet is ejected from the wall into the low speed region of a streak. The point is that the jet is relatively weak with its maximum velocity being 30% of the freestream velocity. A hairpin vortex is generated by the jet injection. The hairpin generates another hairpin vortex and several other small vortices in the vicinity, however all of them gradually weaken as they travel downstream. Then, suddenly, a chain of vortex production starts. Many hairpin vortices and straight longitudinal vortices are generated, which are sometimes interact with each other, and the boundary layer becomes turbulent.

Our current interest is in the local conditions of the flow field which allow the generation of these new vortices. Our ongoing analysis has revealed that the locations where the new vortices are generated coincide with the locations where the streamwise gradients of the streamwise velocities are high. However, the details are yet to be cleared.

References

1. Wygnanski, I., Sokolov, M. and Friedman, D. Jour. of Fluid Mech. Vol.78. 785-819, 1976.
2. Makita, H. and Nishizawa, A. Jour. of Turb. Vol.2. 1-14, 2001
3. Matsubara, M., Alfredsson, P. H. Jour. of Fluid Mech. Vol. 430. 149-168, 2001.

PARALLEL TECHNOLOGIES FOR NUMERICAL SIMULATION OF TURBULENT FLOWS ON VARIOUS COMPUTING ARCHITECTURES

A. Gorobets[1], F.X. Trias, S. Soukov
[1]KIAM RAS, Moscow, cherepock@mail.ru

The work is devoted to the development of efficient parallel algorithms and performing large-scale supercomputer simulations of turbulent flows on different supercomputer architectures. It represents technologies of parallel computations for modeling of CFD problems using high-order finite-volume methods. Different stages are considered: from development of an algorithm, design of software implementation, to computing on large-scale supercomputers.

The parallelization is based on multi-level parallel model that combines different types of parallelism: shared and distributed memory models, single and multiple instruction streams with multiple data streams. MPI is used on the first level to couple computing nodes of a supercomputer. On the second level OpenMP is used to engage multiple CPU cores. The third level exploits the computing potential of massively-parallel accelerators. The hardware independent OpenCL standard is used to compute on accelerators of different architectures.

Technology of performing simulations considers optimal choice of domain geometry, minimization of disk space usage, choice of optimal parallel model configuration, etc. Attention is paid on minimizing queue system waiting time; on the reliability of computations, avoiding crashes of a simulation, automatic correction and optimization of simulation parameters, etc. Ways to improve quality of averaged fields and spectra data are presented.

The recent DNS performed using the high-order finite-volume parallel code based on a symmetry-preserving discretization [1] are to be presented. The cases include a flow around an infinite square cylinder at Reynolds number 22000. The flow has been computed using the 4-th order scheme on a grid with around 300 millions of nodes on MareNostrum supercomputer (BSC, Spain). Another case considered is a natural convection flow inside a closed air-filled differentially heated cavity of height aspect ratio 3.84 and depth aspect ratio 0.86. The Rayleigh number (based on the cavity height) is $Ra = 1.2 \times 10^{11}$. This configuration resembles the experimental set-up performed by D. Saury et al. [2]. The simulation has been carried out on a grid with 100 millions of nodes using MVS-10P supercomputer (JSC RAS, Russia). Finally, DNS of Rayleigh–Bénard convection at $Ra = 10^{10}$ (grid with 600 millions of nodes, 4-th order scheme) is to be presented.

References

1. A. Gorobets, F.X. Trias, A. Oliva, A parallel MPI+OpenMP+OpenCL algorithm for hybrid supercomputations of incompressible flows, Computers and Fluids, 2013,vol. 88, pages 764–772.

2. D. Saury, N. Rouger, F. Djanna, and F. Penot. Natural convection in an air-filled cavity: Experimental results at large Rayleigh numbers. International Communications in Heat and Mass Transfer, 38:679–687, 2011.

METHOD SMIF FOR DNS OF INCOMPRESSIBLE FLUID FLOWS

V. Gushchin[1,2]
[1]ICAD RAS, Moscow, gushchin@icad.org.ru
[2]MIPT (SU), Moscow, gushchin47@mail.ru

Unsteady 3D separated fluid flows are very wide spread phenomena in the nature. The understanding of such flows is very important both from theoretical and from practical points of view. For solving of the Navier-Stokes equations describing 3D incompressible viscous fluid flows the Splitting on physical factors Method for Incompressible Fluid flows (SMIF) with hybrid explicit finite difference scheme (second-order accuracy in space, minimum scheme viscosity and dispersion, capable for work in wide range of Reynolds and Froude numbers and monotonous) based on Modified Central Difference Scheme (MCDS) and Modified Upwind Difference Scheme (MUDS) with special switch condition depending on the velocity sign and the signs of the first and second differences of transferred functions has been developed and successfully applied [1-2]. The Poisson equation for the pressure has been solved by the Preconditioned Conjugate Gradients Method. The parallelization of the algorithm has been made and successfully applied on the massive parallel computers with a distributed memory. For the visualization of the 3D vortex structures in the fluid flows the isosurfaces of β have been drawing, where β is the imaginary part of the complex-conjugate eigen-values of the velocity gradient tensor. The numerical method SMIF has been successfully applied for solving of the different problems: 2D and 3D separated homogeneous and stratified fluid flows around a sphere and a circular cylinder [3-9]; the flows with free surface including regimes with broken surface wave; the air, heat and mass transfer in the clean rooms.

Some numerical results and comparison with experimental data will be demonstrated.

This work has been partly supported by Russian Foundation for Basic Research (grants No. 14-01-00428, 15-51-50023), by the program of the Presidium of RAS No. 8 and by the program No. 3 of the Department of Mathematical Sciences of RAS.

References

1. Belotserkovskii, O.M., Gushchin, V.A., Konshin, V.N. USSR Computational Mathematics and Mathematical Physics V. 27. No. 2. 181-196, 1987.
2. Gushchin, V.A., Konshin, V.N. J. Comput. & Fluids V. 21. No. 3. 345-353, 1992.
3. Gushchin, V.A., Kostomarov, A.V., Matyushin, P.V. Jour. of visualization V. 7. No. 2. 143-150, 2004.
4. Baydulov, V.G., Matyushin, P.V., Chashechkin, Yu.D. Doklady Physics V. 50. No. 4. 195-199, 2005.
5. Gushchin, V.A., Matyushin, P.V. Fluid Dynamics V. 41. No. 5. 795-809, 2006.
6. Baydulov, V.G., Matyushin, P.V., Chashechkin, Yu.D. Fluid Dyn. V. 42. No. 2. 255–267, 2007.
7. Gushchin, V.A., Matyushin, P.V. Comput. Math. and Math. Physics V. 51. No. 2. 251-263, 2011.
8. Gushchin, V.A., Matyushin, P.V. Lecture Notes in Computer Science V. 8236. 311-318, 2013.
9. Gushchin, V.A., Matyushin, P.V. AIP Conf. Proc. V. 1631. 122-134, 2014.

DIRECT NUMERICAL SIMULATION OF HIGH REYNOLDS NUMBER TURBULENCE BY THE K COMPUTER

T. Ishihara[1], K.Morishita[2], M.Yokokawa[3], A.Uno[4], Y.Kaneda[5]
[1]Nagoya University, JST CREST, Nagoya, Japan, ishihara@cse.nagoya-u.ac.jp
[2]Kobe University, Kobe, Japan, morishita@port.kobe-u.ac.jp
[3]Kobe University, Kobe, Japan, yokokawa@port.kobe-u.ac.jp
[4]RIKEN, Kobe, Japan, uno@riken.jp
[5]Aichi Institute of Technology, Toyota, Japan, ykaneda@aitech.ac.jp

Realistic flow phenomena in nature and in industrial applications, such as flows in clouds and high-speed flows past a body, are usually highly non-linear and have huge degrees of freedom. One of the most important parameters in such flows is the Reynolds number ($Re=u'L/v$), where u' is the rms velocity, v is the cinematic viscosity and L is a typical length scale of energy containing eddies in the flow. According to the Kolmogorov theory (K41), the small-scale statistics are universal in sufficiently high *Re* turbulence. Verification of K41 based on the Navier-Stokes equations is the first step for understanding high *Re* turbulence. Recent development of supercomputers enables us to perform large-scale numerical simulations with huge degrees of freedom. Therefore, one may expect that the direct numerical simulation (DNS) of realistic high *Re* turbulence may be not unrealistic in the near future. Recently we have developed a Fourier-spectral DNS code that is optimized for the K computer by using two-dimensional domain decomposition, and have performed a series of large-scale DNS of the incompressible forced turbulence using the K computer. The number of the grid points is up to 12288^3 and the maximum value of *Re* attained by the DNS of turbulence exceeds 10^5 for the first time. The statistics and structures of high *Re* turbulence can be studied directly by the analysis and visualization of such DNS data.

Our previous DNS with the number of grid points up to 4096^3 showed that the energy spectrum has a slope steeper than -5/3 (the Kolmogorov scaling law) by factor 0.1 at the wave number range ($k\eta < 0.03$)[1]. Here η is the Kolmogorov length scale. In the present larger-scale DNS it is shown that the energy spectra with different Reynolds numbers ($Re > 10^4$) are well normalized not by the integral length-scale but by the Kolmogorov length scale, at the wavenumber range of the steep slope. This result indicates that the steep slope is not inherent property in the inertial sub-range, and is affected by viscosity. As for the intermittency of high *Re* turbulence ($Re > 10^4$), visualization shows that the vortical clusters, which consist of strong micro-scale vortices, are thin in one-direction, and have sharper interfaces when the value of *Re* increases.

References

1. Kaneda, Y., Ishihara, T., Yokokawa, M., Itakura, K., Uno, A.: Physics of Fluids, 15. L21-L24, 2003.

INTRODUCTION OF SUPERCOMPUTERS IN NAGOYA UNIVERSITY

K. Ishii
Information Technology Center, Nagoya University, Nagoya, ishii@cc.nagoya-u.ac.jp

The Information Technology Center (ITC) of Nagoya University was originally established as the Computing Center of Nagoya University in 1971, which was reorganized in 2001. The first supercomputer of Nagoya University was Fujitsu supercomputer VP100 installed in the Institute of Plasma Physics in January 1984, which was the first operated supercomputer in Japan. In the following year ITC started the operation of the Fujitsu supercomputer VP200. Since then ITC has been the center of supercomputer in Nagoya University and Nagoya region. ITC is also a core organization of the "Joint Usage/Research Center for Interdisciplinary Large-Scale Information Infrastructures" project, and a part of HPCI (the High-Performance Computing Infrastructure) operated by the Japanese Government, which consists of K-computer center, Nagoya Univ., the university of Tokyo, Kyushu Univ.(Fujitsu SC), the earth simulator, Tohoku Univ., Osaka Univ.(NEC SC), Kyoto Univ.(Cray SC), Hokkaido Univ.(Hitachi SC) etc. The three main missions of ITC as supercomputer center are (i) providing services for supercomputer operations and supporting supercomputer users, (ii) doing research, and (iii) providing education and training.

Currently, ITC is operating three supercomputer systems, a Fujitsu PRIMEHPC FX10 System [384nodes, SPARC64Ixfx(1.650GHz,16core)] at 90.8TFLOPS, and a Fujitsu PRIMERGY CX400/250 [368nodes, Intel IvyBridge (2.7GHz, 12core)] +CX400/270[184nodes, Intel IvyBridge (2.7GHz, 12core) +184XeonPhi3100 (1.1GHz, 57core)] system at 470.6TFLOPS and a SIG UV2000 [1280cores, Intel IvyBridge (2.7GHz, 12core), 20TB shared memory] system and a 7.32PBi strage system.

In April 2015, a Fujitsu PRIMERGY CX400/250 will have 400 nodes. In September, ITC will install a new Fujitsu PRIMEHPC FX100 System [3024nodes, SPARC64XIfx(2.2GHz, 32cores)] at about 3.4PFLOPS instead of FX10.

EXTRACTION OF HIERARCHICAL VORTICES FROM TURBULENT FIELD USING COMPONENT-DEPENDENT FILTER

S. Izawa[1], Y. Kobayashi, Y. Nishio, Y. Fukunishi
[1]Tohoku University, Sendai, izawa@fluid.mech.tohoku.ac.jp

Turbulent vortex motions are discussed by extracting hierarchical vortices in a homogeneous isotropic turbulence at $Re_\lambda = 267$ using a band-pass filter. In general, Fourier filter is often used to extract vortices from a turbulent flow. Vincent[1)] investigated the geometric relation between the large scale vortices and small scale vortices to study the roll-up process of smaller scale vortices. Goto[2)] reported that the small scale vortices exist orthogonally in between a pair of large scale vortices. On the other hand, we take one vorticity component and filter it in the plane normal to it, repeating the procedure for other vorticity components. Compared with the conventional filter, the vortices filtered by the present method have a tendency to be slightly longer and thinner, where the aspect ratio increases by 17 ~ 20 % on average. Also, more complicated structures are found in the flow field.

The present filtering technique is applied to the analysis of a turbulent field and the geometric relation between the target vortices and their surrounding vortices of different scales are investigated, focusing on the stretching of the vortices. As a result, it is found that the vortices in the inertial subrange are likely to be stretched by vortices twice the scale, [3)] and these twice-scaled vortices tend to be located orthogonal to the strongly stretched vortex. This means that these larger vortices are responsible for the stretching of smaller vortices by providing the shear field. Besides, the vortices of double scale more than 7 times of its diameter away from a vortex is found to have no influence on the vortex stretching. These tendencies are consentient regardless of the filtering range.

References

1. Vincent, A. and Meneguzzi, M., “The dynamics of vorticity tubes in homogeneous turbulence”, J. Fluid Mech., 258,pp 24-254,1994.
2. Goto, S., “Coherent structures and energy cascade in homogeneous turbulence”, Prog. Theor. Phys. Supplement, 195 ,pp 139-155, 2012.
3. Hirota, M., “An anisotropic filtering method for multiscale decomposition of vorticity fields”, 27th CFD Symposium , A02-1, USB memory(in Japanese), 2013.

.

SIMPLE *A POSTERIORI* LIMITER TOWARDS FLOW SIMULATIONS WITH 64 TIMES HIGHER RESOLUTION

K. Kitamura[1], A. Hashimoto[2]
[1]Yokohama National University, Yokohama, Japan, kitamura@ynu.ac.jp
[2]JAXA, Chofu, Japan, hashimoto.atsushi@jaxa.jp

Simple *a posteriori* slope limiter is proposed. The *a posteriori* limiting approach was originally developed by Clain, S., Diot, S., and Loubère, R. [1] for higher-order flow computations, in which a lower-order accurate method is chosen only when the candidate, higher-order interpolation violates stability criteria, e.g., positivity of density and pressure, and Discrete Maximum Principle (DMP). In contrast with the conventional *a priori* limiters that select the lower-order accuracy when the interpolated values are "likely" to cause oscillations or divergence, the *a posteriori* limiting is activated only when it is actually needed. However, its original paradigm structure is complicated and calls for many iterations within each time step while the optimal order of accuracy is being searched (from higher to lower orders) - The process has been simplified for 2nd order spatial accuracy here.

The 2nd order method requires only limited (typically near dis-continuities), and "unlimited" (at a smooth flow region) values, both available at the same time (i.e., no extra loop is involved). Then, the *a posteriori* method can choose the solution from, or even blend, them. The present, simple *a posteriori* treatment (SMART) limiter, ϕ_{SMART}, is designed such that it tries to employ the unlimited solution (ϕ=1) where and when possible, and smoothly blend the unlimited and limited ($\phi=\phi_{\lim}$) solutions using Gnoffo's auxiliary limiter, ϕ_{G} [2].

$$\boldsymbol{q}_{i,j} = \boldsymbol{q}_i + \phi_{SMART} \nabla \boldsymbol{q}_i \cdot (\boldsymbol{r}_{i,j} - \boldsymbol{r}_i), \qquad \phi_{SMART} = \phi_G + (1-\phi_G)\cdot\phi_{\lim}$$

Here are two numerical examples: one is 1D, and the other is 2D viscous. Results in Figs. 1-2 demonstrate equivalently four times higher resolution achieved in each direction by the SMART limiter (i.e., 16 times in 2D, and possibly 64 times in 3D extension).

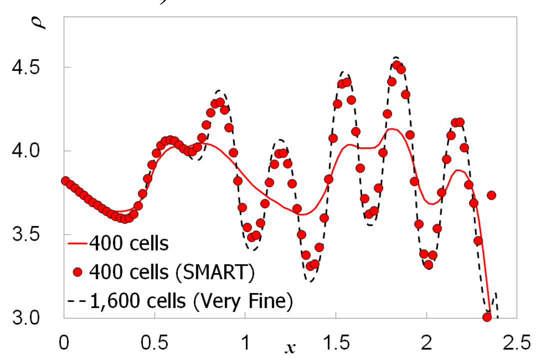


Figure 1. Density profiles for 1D Shu-Osher problem.

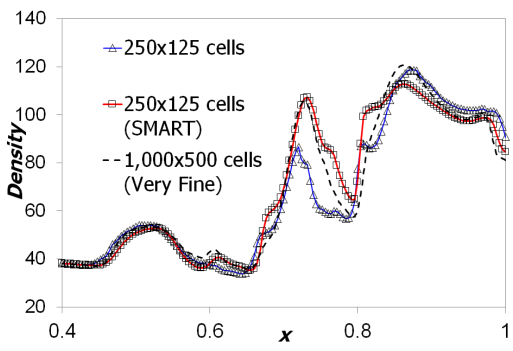


Figure 2. Wall density profiles for 2D viscous shocktube.

References

1. Clain, S., Diot, S., Loubère, R. J. Comput. Phys., Vol. 230, 4028–4050, 2011.
2. Gnoffo, P.A. AIAA 2010-1271, 2010.

NUMERICAL INVESTIGATION OF NONAXISYMMETRIC INSTABILITY GROWTH IN THE 3D BUBBLE-SHOCK INTERACTION PROBLEM

Korneev B. A.[1] and Levchenko V. D.[2]

[1]*MIPT, Moscow,* **boris.korneev@phystech.edu**
[2]*KIAM RAS, Moscow,* **lev@keldysh.ru**

The process of interaction between a spherical bubble and a planar shock [1] is known to have various instable and turbulent features. The phenomenon of the initial rotational symmetry loss appeared in the experiments and numerical simulations is an object of research in this paper. It is shown that small initial nonaxisymmetric perturbations are being increased. The spectral analysis of the instability is made. The amount of detailed fully three-dimensional numerical experiments is held out on the $\mathcal{K}100$ supercomputer in Keldysh Institute. The solver used in this research is based on the RKDG numerical method of solving the fluid dynamics equations [2] and the high-performance DiamondTorre GPU implementation algorithm [3].

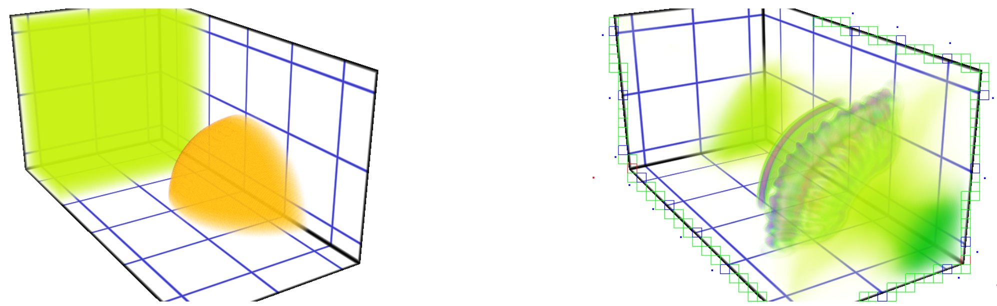


Figure 1: An example of the instability growth. The initial field of density is shown on the left part and the solution at past times is on the right.

The work is partly funded by the RFBR grants 15-01-05052, 12-01-00490 and 12-01-00708, and the "UMNIK" grant of the FASIE.

References

[1] John H.J. Niederhaus, J.A. Greenough, J.G. Oakley, D. Ranjan, M.H. Anderson, and R. Bonazza. *Journal of Fluid Mechanics*, 594:85–124, 2008.

[2] B. Cockburn and C. Shu. Technical report, NASA, 2000.

[3] A.Yu. Perepelkina, V.D. Levchenko, and I.A. Goryachev. In *41st EPS Conference on Plasma Physics*, 2014.

SIMULATION OF REAL AERODYNAMICS PROBLEMS USING HIGHER-ACCURACY LOWER-COST SCHEME ON UNSTRUCTURED MESHES

I.Abalakin, P.Bakhvalov, V.Bobkov, A.Duben, T.Kozubskaya
Keldysh Institute of Applied Mathematics of RAS, Moscow, dualks@gmail.com

In the light of looking for a compromise between high accuracy, robust shock capturing and the corresponding computational costs, which is needed for solving real aerodynamics problems, the talk is focused on the family of vertex-centered EBR (Edge-Based Reconstruction) schemes for unstructured meshes [1] based on quasi-1D edge-oriented reconstruction of variables. These schemes provide high accuracy of the 5th-6th order on translationally symmetric meshes (i.e. linearly deforming uniform Cartesian meshes). Although they possess only the second order of accuracy on arbitrary unstructured meshes, in many cases of coarse meshes normally used in applications, the absolute errors provided by the EBR schemes appear comparable with or even less than the errors of very high order schemes.

To provide efficient shock capturing, we equip the EBR schemes with WENO properties and thereby develop the WENO-EBR schemes. Several test cases verifying these schemes will be presented.

We give the examples of recent CFD and CAA simulations of pre-industrial problems carried out with the use of EBR schemes. In particular, we simulate the high-Reynolds transonic turbulent flow over wedge-shaped body with a backward step, which is typical for aircraft applications, and numerically predict the aerodynamic and aeroacoustic characteristic of helicopter tail rotor.

All the predictions are carried out with the help of in-house code NOISEtte using supercomputers "Lomonosov" of Moscow University Supercomputing Center and "MVS-10P" of Supercomputer Center of RAS. The work on scheme development is supported by Russian Foundation for Basic Research (Project 15-01-07911). The work on turbulent flow simulations is supported by the Russian Scientific Foundation (Project 14-11-00060).

References

1. Ilya Abalakin, Pavel Bakhvalov and Tatiana Kozubskaya. Edge-based reconstruction schemes for prediction of near field flow region in complex aeroacoustic problems, *International Journal of Aeroacoustics,* Vol.13, N 3&4, 2014, p. 207-234

MATHEMATICAL SIMULATION OF TURBULENT SEPARATED TRANSONIC FLOWS AROUND THE AXISYMMETRIC BODIES

I.Yu. Kudryashov[1], A.E. Lutsky[1]

[1]KIAM RAS,Moscow, kudryashov.i.yu@gmail.com, lutsky@kiam.ru

While launching modern spacecrafts, boosters with over-caliber forebody are widely used; their diameter exceeds that of the main airframe. During takeoff, the flow regime changes from subsonic to supersonic.

A numerical investigation of such flows is a very important problem both from theoretical and practical points of view. On the other hand, the flow properties mentioned determine the complexity of the numerical modeling problem and the strict requirements for the algorithms used.

The results of numerical investigation of transonic flow reconstruction occurring with the increase of free stream Mach number are presented. A comparison of the numerical results obtained for the Euler, Navier–Stokes, and the Reynolds–averaged NS equations with the experimental data [1–3] is presented also.

The work has been sponsored by the RFBR (Grants 14-08-00624, 15-51-50023).

References

1. B.N. Dan’kov et al., “Wave Disturbances in Transonic Separation Flows,” Izv. RAN, MZhG, No. 6, 153–165, 2006.
2. B.N. Dan’kov et al., “Peculiarities of Transonic Flow behind Stern Angle Edge of Overcalliper Cone-Cylindrical Body,” Izv. RAN, MZhG, No. 3, 2007.
3. B.N. Dan’kov et al., "The Role of Wave Disturbances in Transonic Separation Flows", TsAGI Science Journal, Vol. XLI №2, pp 19-24, 2010.
4. I.Yu. Kudryashov, A.E. Lutsky, "Mathematical simulation of turbulent separated transonic flows around the bodies of revolution", Mathematical Models and Computer Simulations Vol. 3, No 6,pp 690-696, 2011.

PARALLEL ALGORITHMS FOR SIMULATION OF GAS MIXTURE FLOWS

T. Kudryashova, S. Polyakov
KIAMRAS, Moscow, kudryashova@imamod.ru ,
serge@imamod.ru

In the work we consider gas flows into microchannels of technical systems. The gas is composed of two components: nitrogen and hydrogen. The supersonic expansion of gas mixtures is accompanied by several simultaneous non equilibrium processes: interfusion of layers, shock waves, heat exchange with walls and etc.

We propose method for computing of mixture gas flows. The approach integrates macro method and micro tools. Our macroscopic approach is based on the macroscopic quasigasdynamic equations (QGDM) [1,2] and the correction of the flow parameters is performed by methods of molecular dynamics (MMD). The QGDM system is a generalization of the quasigasdynamic equations for a gas mixture. They are solved by finite volumes method. The system of molecular dynamic equations is used as a sub-grid algorithm. The MMD give a chance to get some information about processes on molecular scale, and on times of a few nanoseconds. Details about our numerical technique is given in [3].

The parallel algorithm is based on "domain decomposition" technique and a splitting on physical processes. Gas dynamics is calculated on CPUs units. The computational area is split into sub-domains. Parallelization is realized on a grid of the processing units. Molecular dynamic calculations are distributed on GPUs. For MMD parallelization an equal number of particles is sent to each computing unit. According to the algorithm the molecular dynamic calculations do not require exchanges.

Parallel computer realization is based on using a cluster or supercomputer with hybrid architecture. Each node of such supercomputer has several multi-core central processors (CPU) and a few graphics processors (GPU). Splitting on physical processes means that QGD-equations are processed by CPUs and MD is processed by GPUs.

The results of calculations obtained in this paper show that the algorithms give high efficiency when implemented on modern hybrid systems with a large number of cores in use.

References

1. T.G. Elizarova, Quasi-gasdynamic equations. Springer, 286 p.p., 2009.
2. G.A.Bird. Molecular Gas Dynamics and the Direct Simulation of Gas Flow. Oxford Science simulations. 1994.
3. S.V. Polyakov, Yu.N. Karamzin, T.A. Kudryashova, O.A. Kosolapov, S.A. Sukov. Hybrid supercomputer platform and development of applications for solving problems of continuum mechanics by grid methods. // News of Russian Southern Federal University. Technical sciences, 2012, № 6 (131), c. 105-115. (in Russian)

SOME ASPECTS OF CALCULATING SHOCKED FLOWS WITH HIGH ACCURACY NUMERICAL SCHEMES

A. M. Lipanov
KIAM RAS, aml35@yandex.ru

The present paper addresses numerical methods and algorithms for the integration in space and computing gradients and second order partial derivatives with respect to spatial coordinates with using variable steps of spatial discretization. These first and second order derivatives are calculated implementing a variable step of integration that is designed in a special manner depending on the quality of the function under consideration. In contrast to the paper [1] where a similar approach is developed to solve numerically the heat conduction equations with the second order of accuracy in spatial coordinates the present paper deals with shocked compressible flows. In particular, we aim to accurately simulate the shock wave intersection phenomenon. We demonstrate that calculation of shocked flows can be performed with the partial derivatives approximations of the second and even higher order of accuracy.

Acknowledgment

The Russian Foundation for Basic Researches (RFBR) is acknowledged for providing the financial support (Grant No 15-51-50023).

References

1. Samarskii A. A. Introduction to Numerical Methods. Moscow, Nauka, 1982, p.272.

SUPERCOMPUTER MATHEMATICAL MODELLING OF MATTER FLOWS IN ACCRETION STELLAR DISKS

A. Lugovsky[1,2], K. Sychugov[1,2]
[1]KIAM RAS, Moscow, alex_lugovsky@mail.ru
[2]NRC "Kurchatov Institute", Moscow, c.kostik@gmail.com

In recent years a large amount of observational data of stellar accretion disks has been received. The mechanism of removing of angular momentum from matter in an accretion disk is the most important factor determining the rate of disk accretion onto a compact object. Reasons of the removal of angular momentum in an accretion disk have long been of interest to researchers all over the world. There are a lot of mechanisms of angular momentum transfer but all these mechanisms meet with some difficulties in attempts to explain the properties of accretion disks.

The results of modeling the structure of gasdynamic flows in stellar accretion disks with gravity accounting are shown. The basic idea of investigation presented in [1] is based on the fact that large-scale vortices play a dominant role in free shear flow. The development of large-scale instability arising in shear flows of stellar accretion disks is investigated. The presence of large-scale structures leads to angular-momentum redistribution in the disk. A new mechanism for defining and characterizing accretion of the matter on the central body is offered.

Methods for the numerical simulation by multiprocessor computers are virtually the only way to complete investigation of such astrophysical objects. The results of the use of MPI technology in the simulation are presented.

Other widespread and well known mechanism of angular momentum transport is the magnetorotational instability (MRI) [2]. The numerical simulations have established that the equilibrium configuration of a gas–dust disk rotating in a spherically symmetrical gravitational potential is subject to the development of strong instability in the presence of a weak magnetic field. It is shown that the development of instability leads to a transport of angular momentum to the disk periphery by large-scale vortex structures, together with the accretion of matter onto the gravitating object.

The created mathematical model is applied to investigate the gasdynamic flow structure in the two-armed global morphology of spiral galaxies. It is shown that the gasdynamic processes lead to appear specific galaxy pattern with hydrodynamic nature and modeling results agree well with the observations.

References

1. O.M. Belotserkovskii,A.M. Oparin andV.M. Chechetkin,Turbulence: NewApproaches (Nauka, Moscow, 2002).
2. Ye.P. Velikhov, Zh. Exp. Teor. Fiz. 36 1399 (1959).

SIMULATION OF SUPERSONIC FLOWS IN THE WING WAKE AND ITS INTERACTION WITH CROSSING SHOCK WAVES.

A.A. Davydov[1] , A.E. Lutsky[1]

[1] Keldysh Institute of Applied Mathematics RAS
125047 Moscow, Russia, lutsky@kiam.ru

The results of supersonic flow in the wing wake and its interaction with shock waves investigation are presented. Numerical simulations have been performed with the help of improved version of 3–order algorithm for the numerical simulation of 3-dimensional turbulent flows with separation. The explicit–implicit LU-SGS method for Navier–Stokes equations and differential equations for the turbulent viscosity has been added. The method allows to reduce the computation time and achieve the efficiency of parallelization more than 80 %. Calculations have been performed on a multiprocessor computer system K-100 in KIAM RAS. Experiments were carried out in a supersonic wind tunnel T–325 (ITAM SB RAS) at the Mach number 3. The rectangular wing with sharp edges has been used to generate wing tip vortex. The system of couple counter wedges with a sharp edge has been used for the formation of shock waves of varying intensity. The results [1-4] were used to estimate the size and topology of the flow pattern and Mach stem. The numerical simulation of the modes with regular and Mach interaction of shock waves has been performed. Wing tip vortex interaction in the case of regular crossing shock waves refers to the so-called weak type. Vortex core passes through a system of shock waves without collapsing and circulation zones formation. The interference of the wake vortex with Mach shock waves leads to destruction of the vortex core, with the formation of the recirculation zone, which represents an almost isobaric region with a low total pressure and Mach number. Inside this region a low-speed toroidal vortex is formed. This mode of interaction is characterized by strong nonstationarity with time–varying dimensions and boundaries of the interaction region.

The work was supported by RFBR projects 15-01-08575, 15-51-50023.

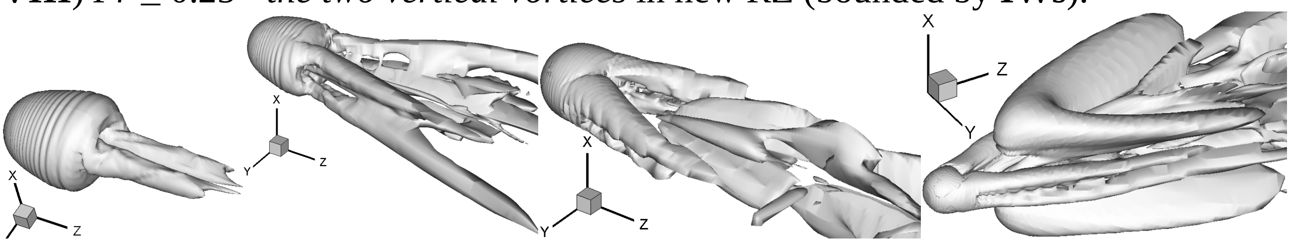
REFERENCES

1. D.J. Azevedo and Ching Shi Liut. Engineering Approach to the Prediction of Shock Patterns in Bounded High-Speed Flows / / AIAA Journal, Vol. 31 , No. 1 , January 1993
2. M. S. Ivanov, G. N. Markelov, A. N. Kudryavtsev, and S. F. Gimelshein. Numerical Analysis of Shock Wave Reflection Transition in Steady Flows / / AIAA Journal, Vol. 36 , No. 11 , November 1998.
3. MS Ivanov , AN Kudryavtsev , SB Nikiforov , DV Khotyanovsky . Transition between regular and Mach -sky reflection of shock waves : new numerical and experimental results / / Aeromechanics and gas-hand speaker , 2002, № 3 , pp. 3-12
4. A. M. Kharitonov, A. E. Lutsky, and A. M. Shevchenko, “Investigations of Supersonic Vortex Cores Above and Behind of a Wing,” in Proceedings of 2nd European Conference for Aerospace Sciences (Brussels, 2007).

TRANSFORMATION OF THE VORTEX STRUCTURE OF THE STRATIFIED FLUID AROUND A MOVING BODY WITH DECREASING OF INTERNAL FROUDE NUMBER

P. Matyushin[1], V. Gushchin[1,2]
[1]ICAD RAS, Moscow, pmatyushin@mail.ru
[2]MIPT (SU), Moscow, gushchin47@mail.ru

At the present paper the density stratified viscous fluid flows around blunt body (sphere with diameter *d*) moving with a velocity *U* have been simulated on the supercomputers of JSCC RAS (www.jscc.ru) on the basis of the Navier-Stokes equations in the Boussinesq approximation with two main non-dimensional parameters: internal Froude number $Fr = U/(N \cdot d)$ and Reynolds number $Re = U\,d/\upsilon$, where *N* is the buoyancy frequency, υ is the kinematical viscosity coefficient. For solving of these equations the numerical method SMIF [1] has been used. For the visualization of the 3D vortex structures in the wake the isosurfaces of β has been drawing, where β is *the imaginary part of the complex-conjugate eigen-values of the velocity gradient tensor* (fig. 1). The length of the internal waves (**IWs**) in the vertical plane is $\lambda/d \approx 2\pi \cdot Fr$ (fig. 1d). The following classification of flow regimes around a sphere at $Re = 250$ and $0 \le Fr \le \infty$ has been obtained by SMIF: **I**) $Fr > 10$ – *two steady threads in the wake* (fig. 1a) [2]; **II**) $5 \le Fr \le 10$ – *two steady threads* with four additional threads connected with vortex sheet surrounding the sphere; **III**) $1.5 \le Fr < 5$ – *two wavy threads* with four additional threads (fig. 1b); **IV**) $0.9 < Fr < 1.5$ - *the non-axisymmetric attached vortex* in recirculation zone (**RZ**) (fig. 1c); **V**) $0.6 < Fr \le 0.9$ – *the two symmetric vortex loops in* RZ [3]; **VI**) $0.4 \le Fr \le 0.6$ – *the absence* of RZ; **VII**) $0.25 < Fr < 0.4$ – *a new RZ*; **VIII**) $Fr \le 0.25$ - *the two vertical vortices* in new RZ (bounded by **IWs**).



a) $Fr = 100$, $Re = 250$; b) $Fr = 2$, $Re = 250$; c) $Fr = 1$, $Re = 250$; d) $Fr = 1$, $Re = 200$

Fig. 1. Vortex structures of the viscous fluid flows around a sphere: a) two threads, b) six threads; c-d) four threads (β = 0.04, 0.04, 0.08, 0.02).

This work has been partly supported by Russian Foundation for Basic Research (grants No. 14-01-00428, 15-51-50023), by the program of the Presidium of RAS No. 8 and by the program No. 3 of the Department of Mathematical Sciences of RAS.

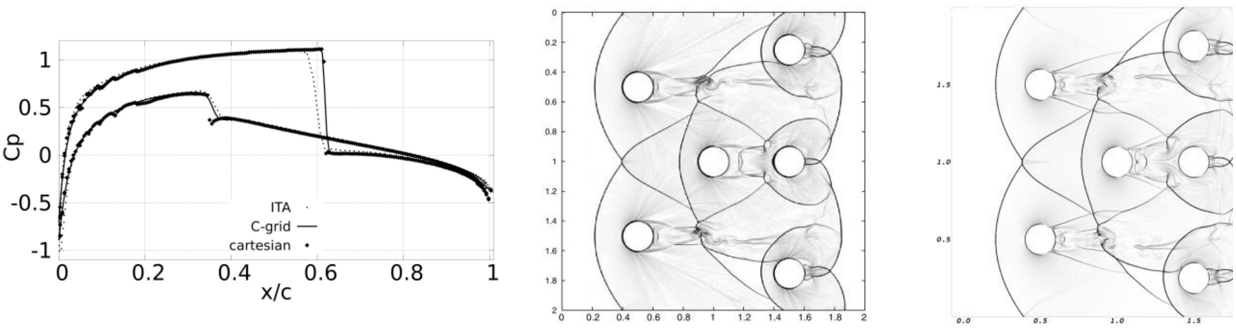
References

1. Belotserkovskii, O.M., Gushchin, V.A., Konshin, V.N. USSR Computational Mathematics and Mathematical Physics V. 27. No. 2. 181-196, 1987.
2. Gushchin, V.A., Matyushin, P.V. Fluid Dynamics V. 41. No. 5. 795-809, 2006.
3. Gushchin, V.A., Matyushin, P.V. Comput. Math. and Math. Physics V. 51. No. 2. 251-263, 2011.

SUPERCOMPUTER MODELING COMPRESSIBLE FLOWS WITH A CARTESIAN GRID METHOD

I. Menshov[1], P. Pavlukhin[2]
[1]*KIAM RAS, Moscow, menshov@kiam.ru*
[2]*Research Institute "KVANT", Moscow, giperchuv@mail.ru*

We propose a Cartesian-grid numerical method that allows one to perform calculations of gas dynamics equations in complex geometry domains by using simple geometry non-fitted Cartesian grids. This approach possesses an important feature of algorithm homogeneity (computational primitivism) which makes it quite suitable for massive parallel architectures (e.g., multi-GPU computer systems). The method steams from an alternative mathematical formulation of the boundary value problem where boundary conditions are modeled by a so-called compensation flux – a surface-defined function introduced in the right-hand side of the governing equations. It is shown that the compensation flux can be chosen in such a way so that the solution to the modified equations in the whole space projected onto the problem domain coincides with the solution to the initial boundary value problem [1]. The numerical method for solving the modified equations is hybrid explicit-implicit [2] based on the Godunov approach. The system of discrete equations is solved with the LU-SGS approximate factorization method. We also describe an effective multi-GPU two-levels parallel realization based on executing staggered (chess-type) counting loops. Fig. 1(a) illustrates comparison between numerical solutions for the NACA0012 airfoil flow obtained with a body fitted structured grid and a Cartesian grid, respectively; figs. 1(b) and 1(c) - comparison of the presented method with an alternative one [3].



(a) (b) (c)

Fig.1. (a) Cp distribution, NACA0012, M=0.8, α=1.25°; solid line- body fitted grid , markers —Cartesian grid. Flow around a set of cylinders, M=3: present (b) и alternative [3] (c).

This work was partly supported by the Russian Foundation for Basic Researches (RFBR) (Grant No 15-51-50023).

References

1. I.S. Menshov, M.A. Kornev // Math. Mod. and Comp. Simul. Vol. 6(6). 612-621, 2014.
2. I. Menshov, Y. Nakamura // AIAA Jour. Vol. 42(3). 551-559, 2004.
3. O. Boiron, G. Chiavassa, R. Donat // Computers & Fluids. Vol. 38. 703-714, 2009.

CLUSTERING AND ENTROPY GROWTH OF QUASI-GEOSTROPHIC POINT VORTICES UNDER PERIODIC BOUNDARY CONDITIONS

T. Miyazaki[1], Y. Shimoda[2], Y. Konno[3]
[1]***Univ. Electro-Communications, Tokyo, miyazaki@mce.uec.ac.jp***
[2]***Univ. Electro-Communications, Tokyo, shimodai@miyazaki,mce.uec.ac.jp***
[3]***Univ. Electro-Communications, Tokyo, k1112057@edu.cc.uec.ac.jp***

The statistical mechanics of quasi-geostrophic point vortices of mixed sign is investigated numerically and theoretically, extending the previous work by our group (Funakoshi *et al*., 2012). Direct numerical simulations of a point vortex system under periodic boundary conditions are performed using a fast special-purpose computer for molecular dynamics (GRAPE9). Clustering of point vortices of like sign is observed and two tall columnar vortices appear in the course of time. These numerical results are analyzed quantitatively using a density-based algorithm. The number of clusters decreases as about t^{-1}, which is significantly slower than $t^{-1.25}$ found in the previous spectral simulations of geostrophic turbulence (e.g. McWilliams *et al*., 1999). The evolution of the cluster size and shape is also studied in detail. The equilibrium states, formed in the later stage of simulations, are the sn-sn dipole solutions of the two-dimensional mean field equation, i.e., the sinh-Poisson equation. Funakoshi *et al*. derived a three-dimensional mean field equation based on the maximum entropy theory. They presented several branches of three-dimensional solution. The three-dimensional branch was found to have smaller entropy value compared with the two-dimensional sn-sn dipole branch. We find that the entropy increases monotonically and asymptotes to the value of the two-dimensional sn-sn dipole branch in the numerical simulations.

References

Funakoshi S, Sato T and Miyazaki T: Maximum entropy state of the quasi-geostrophic bi-disperse point vortex system: Bifurcation phenomena under periodic boundary conditions, Fluid Dyn. Res. **44** 031407 (13pp), 2012
McWilliams J C, Weiss J B and Yavneh I: The vortices of homogeneous geostrophic turbulence, J. Fluid Mech. **401** 1-26, 1999

MHD HEAT SHIELD IN ARGON ARCJET PLASMA FLOW

Y. Nagata[1], K. Yamada[2], T. Abe[3], S. Yanase[4]
[1]Okayama University, Okayama, ynagata@okayama-u.ac.jp
[2]JAXA/ISAS, Sagamihara, yamada.kazuhiko@jaxa.jp
[3]JAXA/ISAS, Sagamihara, tabe@gd.isas.jaxa.jp
[4]Okayama University, Okayama, yanase@okayama-u.ac.jp

Magneto-Hydro Dynamics (MHD) flow control technology has a potential to break through to atmospheric reentry vehicle system design. Reentry vehicles, such as space shuttle and Apollo capsule, are exposed to severe thermal environment due to very high aerodynamic heating when it returns from space to ground. During the atmospheric entry flight at very high speed over 25 in Mach number, weakly ionized plasma flow is generated owing to the strong bow shock ahead of the vehicle. Since 1950s, the MHD heat shield was suggested to ease the thermal environment[1,2]. As the magnetic field is applied around the vehicle, the induce current is generated and the Lorentz force acts against the plasma flow because of the MHD interaction. As a result of this interaction, the flow field could be changed and some effects are obtained; the shock layer enhancement, the convective heat flux reduction, and the drug force enhancement due to the reaction force of the Lorentz force. This type of flow field contains some phenomena; hypersonic, thermal and chemical non-equilibrium, MHD interaction. However, only a few experimental data are available to validate. Then we investigate this problem numerically and experimentally.

In the MHD flow control, the magnetic field configuration is a controllable parameter, and then, the magnetic field can be inclined from the body axis. This tilted magnetic field condition should cause the 3-dimentional flow field and the magneto-aerodynamic force deflection: not only the drug force but also the side force. This force deflection is confirmed by the Argon arcjet wind tunnel experiment[3]. Then, we performed the MHD simulation for this experiment to validate. In the present calculation, 3-dimentional MHD equations and simplified Maxwell equations based on the low magnetic Reynolds number assumption are employed because the magnetic Reynold number based on the maximum magnetic field strength is equal to unity. Then, the induction magnetic field is not considered.

As a result, the simulation is similar to the experiment if the artificial insulating boundary is set to assume the finite plume size. In contrast, they disagree on the side force direction in the case of incoming uniform plasma flow. In the experiment, the arcjet plume size is relatively small to the body size. Thus, like the drag force, the side force could be substantially affected by the insulating boundary location.

References

1. Ziemer, R. W. and Bush, W. B.: Phys. Rev. Lett. Vol. 1. No 2. 58-59, 1958.
2. Poggie, J. and Gaitonde, D. V.: Phys. Fluids. Vol. 14. No 5. 1720-1731, 2002.
3. Kawamura, M., Nagata, Y., Katsurayama, H., Otsu, H., Yamada, K., and Abe, T.: J. Spacecraft. Vol. 50. No 2. 347-351, 2013.

DETERMINATION OF MATERIALS MACROPARAMETERS FOR TECHNICAL MICROSYSTEMS

V. Podryga[1], S. Polyakov[1]
[1]KIAMRAS, Moscow, pvictoria@list.ru, polyakov@imamod.ru

The present stage of science development is characterized by a large number of researches carried out for the purposes of elaboration and implementation of nanotechnology into production. Nanotechnologies suppose a good knowledge of the properties of materials and mediums used in industrial processes. One way to obtain information about the medium properties is a molecular modeling.

In this paper it is talking about the properties of gas medium which are used as transport systems in installations of supersonic gasdynamic sputtering of nanoparticles on the substrates [1]. Modern computer technology allows to calculate the properties of the gas medium at the molecular level. This gives the possibility for determining the gases macroparameters and constants of their interaction with the sputtering installations metal surfaces.

In operation by means of a molecular dynamics method [2] determination of real gas (nitrogen) near a metal (nickel) surface macroparameters is made. From molecular dynamics calculations the equation of state for pressure and energy (i.e. the compressibility factor and heat capacity), viscosity, enthalpy, Gibbs energy, and entropy are defined. In addition for the subsequent simulation of gas-metal system the equilibrium parameters of nickel are calculated, such as temperature, pressure, coefficient of thermal expansion, compressibility factor, diffusion factor and heat conduction.

Common numerical approach is presented in [1], and it relies heavily on supercomputing. In this paper the main attention is paid to improving the efficiency of parallel computing and increase the reliability of the simulation results. As a result of the calculation program optimization we can now analyze the properties of systems containing 10 million particles and more. In the carried-out calculations of such systems, besides the above mentioned parameters of the gas medium, we were able to calculate in details the process of nitrogen on the nickel surface adsorption. The obtained results agree well with theoretical data.

References

1. Karamzin Yu., Kudryashova T., Podryga V. and Polyakov S. Numerical Simulation of the Gas Mixture Flows on Hybrid Computer Systems. Proceedings of The Ninth International Conference on Engineering Computational Technology, P. Ivanyi and B.H.V. Topping (Editors), Civil-Comp Press, Stirlingshire, Scotland, paper 28, 2014. doi:10.4203/ccp.105.28
2. D.C. Rapoport. The Art of Molecular Dynamics Simulations. Second Edition, Cambridge University Press, 2004.

PROBLEM OF DATA VISUALIZATION FOR MOLECULAR DYNAMICS SIMULATION OF GAS-METAL MICROSYSTEM

D. Puzyrkov[1], V. Podryga[1] , S. Polyakov[1]
[1]KIAM RAS, Moscow, dpuzyrkov@gmail.com, polyakov@imamod.ru

Evolution of the computer technology and an increasing its performance make it possible to calculate the properties of complex systems at the molecular level. The mathematical models, which describe similar processes, can contain information about a large quantity of particles up to several billion. Such large volumes of data and examination of strongly nonlinear effects lead to the problem of the representation of simulation results. One of the methods of representation consists in visualization of trajectories and states of all particles in the region.

In the work [1] the interaction of gas flow with the metallic plate, which is provided by the methods of molecular dynamics, is examined. And if the numerical algorithm of calculation was completely investigated and approved by us, then the analysis of the results of simulation became ever more complicated in view of an increase in the size of the system being simulated. In connection with that we paid special attention to the problem of processing and visualization of the results of calculations. For its solution we proposed to represent on the screen both the position of particles and the vectors of their speeds at the assigned moments of time and the trajectory of separate particles. The most interesting occurs on the border of the gas-metal (on the surface of metal plate). That's why the selected method of visualization must consider the layer's structure of metal and gas located above it.

To achieve required visualization result a parallel, user configurable, server software was created. The software makes it possible to render the large volumes of data in the appropriate layers of system and the recording as a result video file with the necessary information. As the set of instruments for the solution of the visualization problem was used “enthought mayavi” [2], namely its subset “mlab”, which is been expansion for VTK. The mlab allows to make offscreen-rendering of vtk scene with any volume of data, under the condition that it fits the RAM. For using this program in our goals it was necessary to develop additional scripts for the visualization of data points (such as molecules and atoms), and also animation of rotations and zoom of the scenes. As a result, we succeeded in examining well interaction of the molecules of gas with the metal, for example, the effect of adsorption.

References

1. Podryga V.O., Polyakov S.V. Molecular simulation of interaction of gas mixture with metal surface. Mesh methods for boundary-value problems and applications. Proceedings of 10th International Conference. Kazan: Otechestvo, 496-502, 2014.
2. Mayavi documentation: http://docs.enthought.com/mayavi/mayavi/mlab.html

NUMERICAL MODELLING OF HIGH SPEED REACTING FLOWS ON MULTIPROCESSOR COMPUTERS

I. Semenov
Institute for Computer Aided Design RAS, Moscow, semenov@icad.org.ru

The present paper deals with the numerical investigation of high speed reacting flows in gaseous and heterogeneous mixtures. The two problems were considered. The first problem is detonation initiation and propagation in channel or tube which fulfilled reacting mixture such as methane-air. It’s well known that detonation structure in methane-air mixture is highly irregular. The numerical investigation of detonation initiation and propagation in methane-air required significant computational cost due to the large detonation cell size that should be resolved appropriately. Our approach is based on the set of equations for 2D and 3D transient flow of inviscid, compressible, multicomponent, explosive gaseous mixture. The numerical procedure for solving Euler equations is based on the finite volume approach, explicit time integration, interpolation procedure to enhance the spatial accuracy of the scheme. The fluxes through the computational cells faces are calculated with Godunov's method. Reduced chemical kinetics models are used for describing reactions in the mixture. Fig.1 illustrates “numerical soot footprints” (gas pressure maximums distributions) in planar channel which reveal highly irregular detonation structure.

The second problem is lifting and dispersing of a dust layer behind the propagating shock wave as well as ignition, combustion of coal particles and dust-layer detonation formation in tube. The mathematical statement of the problem is based on 2D equations of two-phase, viscous, reactive, compressible flow within a coupled two-velocity and two-temperature formulation. The acceleration of leading shock wave and dust-layered detonation formation are connected with increasing and intensification of combustion zone that strongly depends on arising system of the oblique shock waves due to the development of the dust layer instabilities.

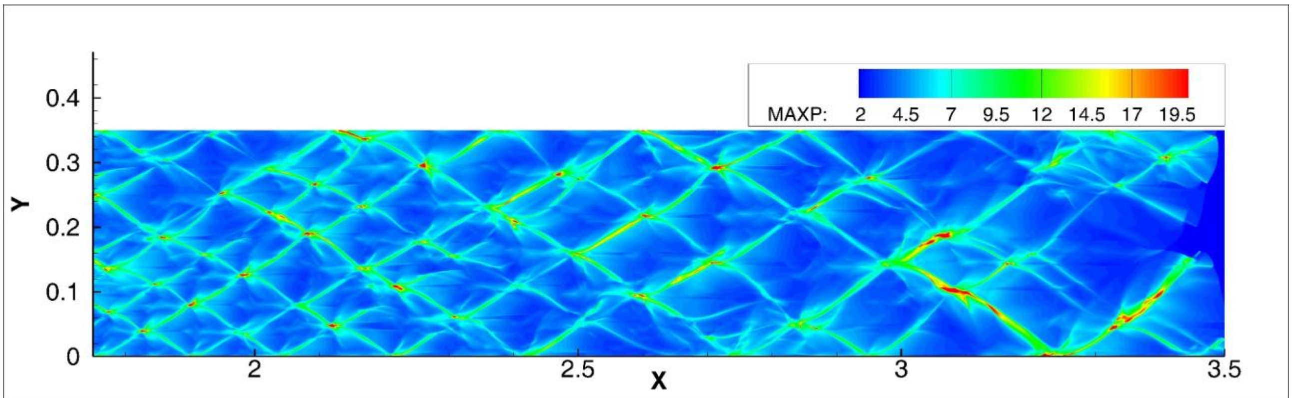


Fig. 1 “Numerical soot footprints” in channel filled with stoichiometric methane-air mixture.

TURBULENT BOUNDARY LAYER FLOW SIMULATION USING ALGEBRAIC EDDY VISCOSITY MODEL AND WALL FUNCTIONS

I. Menshov, A. Severin
KIAM RAS, Moscow, menshov@kiam.ru, severin@kiam.ru

The present paper describes our successful attempts to model separated boundary layer flows using a simple algebraic turbulence model implemented with a special type of boundary conditions written in terms of wall functions (the law of the wall).

The gas flow is simulated using the spatially averaged Navier-Stokes equations with the subgrid turbulence model of Smagorinsky [2]. Near the wall surface turbulent eddy viscosity is determined by the characteristic length that is defined by the distance from the wall in its vicinity and by a typical computational cell size outsize that region.

We impose special boundary conditions on the body surface, which are constructed on the base of the logarithmic velocity profile known as the law of the wall. The method of wall functions that we propose in this paper involves a special way of approximating the velocity in the layer of adjacent to the surface cells. We use a special subcell velocity reconstruction near the surface, which is derived from the wall functions (the law of the wall). Using this reconstruction, the derivative in the normal direction of the tangential component of the velocity vector is calculated, and the wall stress is then calculated by this derivative. The calculated stress is implemented in the finite-volume discretization to approximate the viscous flux at the body surface. A similar approach is considered in the paper [3]. The baseline numerical method for solving the Navier-Stokes equations we employ is described in [1].

Implementing the Smagorinsky eddy viscosity and proposed wall functions allow us achieve reasonable accuracy in modeling turbulent boundary layer flows within the LES approach applied in the whole domain without additional parameters commonly used in RANS methods. Moreover, it works with just the same computational grids as RANS methods do. These are demonstrated on examples of calculation turbulent boundary layer flows for a flat plate and a ONERAM6 wing.

Calculations have been carried out on the supercomputer “Chebyshev” of the Lomonosov Moscow State University.

References

1. I. Men’shov, Y. Nakamura, Hybrid Explicit–Implicit, Unconditionally Stable Scheme for Unsteady Compressible Flows // AIAA Journal, Vol. 42, No. 3, pp. 551-559, 2004.
2. Joseph Smagorinsky. General Circulation Experiments with the Primitive Equations. Monthly Weather Review, 1963.Vol. 91, pp. 99-164.
3. T. Knopp. On grid-independence of RANS predictions for aerodynamic flows using model-consistent universal wall-functions // European Conference on Computational Fluid Dynamics ECCOMAS CFD, 2006.

VORTICAL FLOWS IN THE AORTA AND THEIR RELATIONS TO GEOMETRICAL CHARACTERISTICS

H. Suito
Okayama University, Okayama, Japan, suito@okayama-u.ac.jp

In this paper, vortical flow structures in the thoracic aorta are presented as they relate to aortic aneurysms. Thoracic aortic aneurysm is one of the life-threatening diseases, which slowly grows with advancing age of the patient and may be at a risk of rupture. Many papers have reported on the risk factors, however, natural history of the development of an aneurysm has not been fully understood[1, 2].

There are so many parameters characterizing the blood flow, for example, geometric, kinematic and physiologic parameters. The information on which parameter would be the most important to predict for the generation and development of the aneurysms should be useful for clinical medicine. In this study, we draw attention to geometrical characteristics of the blood vessels, because the differences in the geometry of the blood vessels bring about differences in the vortex structures in the blood flow, which cause different wall shear stress (WSS) and oscillatory shear index (OSI) distributions.

References

1. E. Isselbacher, "Thoracic and abdominal aortic aneurysms", Circulation, **111**, pp. 816–28, 2005.
2. J. Elefteriades, "Natural history of thoracic aortic aneurysms: indications for surgery, and surgical versus nonsurgical risks", *Annals of Thoracic Surgery*, **74**, S1877–1880, 2002.
3. K. Takizawa and T.E. Tezduyar, "Multiscale space–time fluid–structure interaction techniques", *Computational Mechanics*, **248**(3), 247–267, (2011).
4. K. Takizawa, T.E. Tezduyar, A. Buscher and S. Asada, "Space–time interface-tracking with topology change (ST-TC)", *Computational Mechanics*, **54**(4), 955–971, (2013).
5. K. Takizawa, T.E. Tezduyar, A. Buscher and S. Asada, "Space–time fluid mechanics computation of heart valve models", *Computational Mechanics*, **54**(4), 973–986, (2014).
6. H. Suito, K. Takizawa, V.Q. Huynh, D. Sze, and T. Ueda, "FSI analysis of the blood flow and geometrical characteristics in the thoracic aorta", *Computational Mechanics*, **54**(4), 1035–1045, (2014).

Drag exerted on a spherical particle by a slow motion of a rarefied gas

Satoshi Taguchi
Department of Mechanical Engineering and Intelligent Systems, The University of Electro-Communications,1–5–1, Chofugaoka, Chofu, Tokyo 182-8585, Japan
taguchi.satoshi@uec.ac.jp

A slow uniform flow of a rarefied gas past a sphere with a uniform temperature is considered. The steady behavior of the gas is investigated on the basis of the Boltzmann equation by a systematic asymptotic analysis for small Mach numbers in the case where the Knudsen number is finite. Introducing a slowly varying solution whose length scale of variation is much longer than the sphere dimension, the fluid-dynamic-type equations describing the overall behavior of the gas in the far region are derived. Then, the solution in the near region which varies on the scale of the sphere size, described by the linearized Boltzmann equation, and the solution in the far region, described by the fluid-dynamic-type equations, are sought in the form of a Mach number expansion up to the second order in a way that they are joined in the intermediate overlapping region. As a result, the drag is derived up to the second order of the Mach number, which formally extends the linear drag obtained by Takata et al. [1] to a weakly nonlinear case. Numerical results for the drag on the basis of the Bhatnagar-Gross-Krook (BGK) model are also presented.

This work was supported by JSPS KAKENHI Grant Number 25820041.

REFERENCES

1. S. Takata, Y. Sone, and K. Aoki, “Numerical analysis of a uniform flow of a rarefied gas past a sphere on the basis of the Boltzmann equation for hard-sphere molecules", *Phys. Fluids A* **5**, 716–737 (1993).

Authors index

Notes

Notes