Scientific Visualization, 2023, volume 15, number 4, pages 77 - 91, DOI: 10.26583/sv.15.4.07

Identification of Multiscale Vortex Structures and Transition Fluid Features in CFD and Numerical Astrophysics

V. D. Goryachev[1]

Tver State Technical University

[1] ORCID: 0000-0003-0645-3628, gdv.vdg@yandex.ru

Abstract

The need for effective postprocessor analysis of the results of extensive calculations on supercomputers remains extremely high when using new complex computing systems in modeling super- and hypersonic gas flows, predictive modeling of heat and mass transfer processes in jet engines and new energy devices, in solving multiscale problems for astrophysical simulation. Requirements for improving the quality and versatility of new software and visualization codes are increasing with the transition to exascale computing, which requires the improvement of new service systems, in particular systems for visualizing and analyzing the results of multi-scale calculations with dramatically increased input and output data. This paper considers the experience of using an authorized visualization system in the practice of solving some tasks with vortex flow structures and unpredictable instability.

Keywords: visualization, computational fluid dynamics, vortex structures, large data.

1. Introduction

A significant increase in the computing power of modern supercomputers with a sharply growing number of processors stimulates increased requirements for the quality and functionality of modeling by CFD methods when solving problems in plasma physics, gasdynamics and astrophysics tasks in a wide time and space range. The enormous volume of post-computation data requires fast, high-quality 3D visualization to properly analyze rapidly growing sets of exascale computational results in depth. These outputs must be processed “in-situ” on computer servers, or remote workstations using constantly improved graphical post-processing codes. Advanced software for visualizing numerical fluid dynamics and astrophysical simulation results must be able to comprehensively display all characteristics of time-dependent result sets, both real fields and pseudo-real derived fields, often using photorealistic rendering for expressiveness of flow features.

As fairly common visualization programs for processing the results of hydro- and gas dynamics simulation in physics applications, we can mention the VisIt, ParaView, Ensight programs and some another packages for visualization in interdisciplinary research. These post-processors use the open source VTK rendering toolkit [1], an object-oriented library for rendering 3D data, and support a wide range of rendering algorithms, including scalar, vector, tensor, texture, and volumetric representation methods. They use advanced modeling techniques such as implicit modeling, polygon reduction, mesh smoothing, cutting, contouring, etc. The rendering toolkit adds a rendering abstraction layer on top of the underlying graphics library. This higher level simplifies the task of creating users visualization scenes. The list of special software for graphical analysis of computer simulation results is continuously updated. As a rule author's in-house made and commercial programs are written in C, C++, Java and Python using different frameworks.

In domestic research on numerical fluid dynamics, both proprietary graphic postprocessors of own design and commercial visualization systems are used. The methods used quite conservatively repeat those obtained in experimental visualization, but are continuously supplemented with new approaches for the representation and transformation of scalar, vector and tensor fields, in particular, in computational gas dynamics problems. The classification and features of such methods are given in representative review papers [2-4]. Using a variety of libraries created by various authors and groups is a common way to create custom rendering systems that are tailored to the computer environment used by the code developers. An example is the software visualization package RemoteView that is developed for a supercomputer cluster of IMM RAS [5].

When developing programs for visualizing computer systems in astrophysics, an important condition is to take into account the gigantic dimensions of simulated cosmic spaces, identifying both significantly extended spatial zones of galaxies or nebulae, and details of much smaller (on the appropriate scale) objects, for example, individual stars or planets [6]. Astrophysical scientific visualization software can provide pseudo-realistic visualization of cosmic fields and comprehensively display all the features of a simulation dataset. Users of the programs are encouraged to improve the stylistic solution for visualizing scenes adopted in experimental astronomy (photos of processes in the Universe), and pseudo-color illustrations for scientific publications on a specific task are quite informative. Recent codes have seen significant increases in system processing efficiency of data output, taking into account the parallelization of rendering operations using of huge set CPU processors and GPU, and the improvement of data input-output operations in the remote data processing. As an example of advanced visualization systems in breakthrough research in computational astrophysics can be note a program NVIDIA IndeX [7], a rendering and visualization tool for high performance computation. The program utilizes GPU clusters for the scalable, real-time visualization of large datasets and compute capacity by utilizing all distributed resources of multi-GPU environments on supercomputers for visualizing virtually unlimited dataset sizes. The postprocessing system produces high fidelity and visual accurate visualizations, which was demonstrated in treatment of simulation results in challenge projects such "Galactic Winds", using simulation framework Computational Hydrodynamics on Parallel Architectures (CHOLLA) [8].

2. Visualization in multi-physical CFD simulation

As part of some CFD projects, a visualization and interpretation system was developed for processing the results of solving large-scale modeling problems in CFD applications, numerical astrophysics, gas dynamics and modeling of energy processes.

High Definition Visualization and Interpretation System - HDVIS (first mention in [9]) is designed as a multi-platform Java graphics computing environment. The system software is written in Java using the IntelliJ IDEA - development environment for Java SE and JDK-JEP (JDK Enhancement Proposals). The graphical user interface of the system is created using the Java Swing and Java OpenGL (JOGL) libraries. Users of the system interact with big data output by manipulating visual widgets that provide interaction appropriate to the type of data and can render high-resolution visual scenes using additional tools to enhance quality of analysis. The functioning of a multiplatform visualization system is possible in Linux, macOS, Windows and other operating systems, which are often used on workstations and servers of high-performance computers. In terms of its main parameters, developed system is quite competitive when compared with other developments of CFD visualization systems, where data are specified on block-structured grids with four or more billion nodes. A comparison of the capabilities of the developed visualization system and commonly used graphic post-processors such as VisIt [10] and Tecplot [11], used on middle-class graphics stations, showed a number of advantages of the presented in-house system.

Visual post-processing in HDVIS can be thought of as a pipeline consisting of conventional actions: input and filtering of output data and its transformation into another view by changing its contents through the generation of new objects construction of a visual scene with the combination of real-like or partially abstract objects - scalar fields, surface set, pathlines, streak lines, vortex lines, etc. [12]. Inheritance of various operations in a graphical menu with a drop-down tree reflects user actions in the visualization system. The project windows contain a menu for managing the contents of the scene with a tree of connections of related variables, both source data and derived values arrays, hierarchically created and archived. The generated arrays of variables are added to the visualization database or temporarily used in the project. Working within HDVIS, the user sequentially goes through the various stages of creating visualization scenes in accordance with his plan.

Windows of system with some menu are shown in Figure 1.

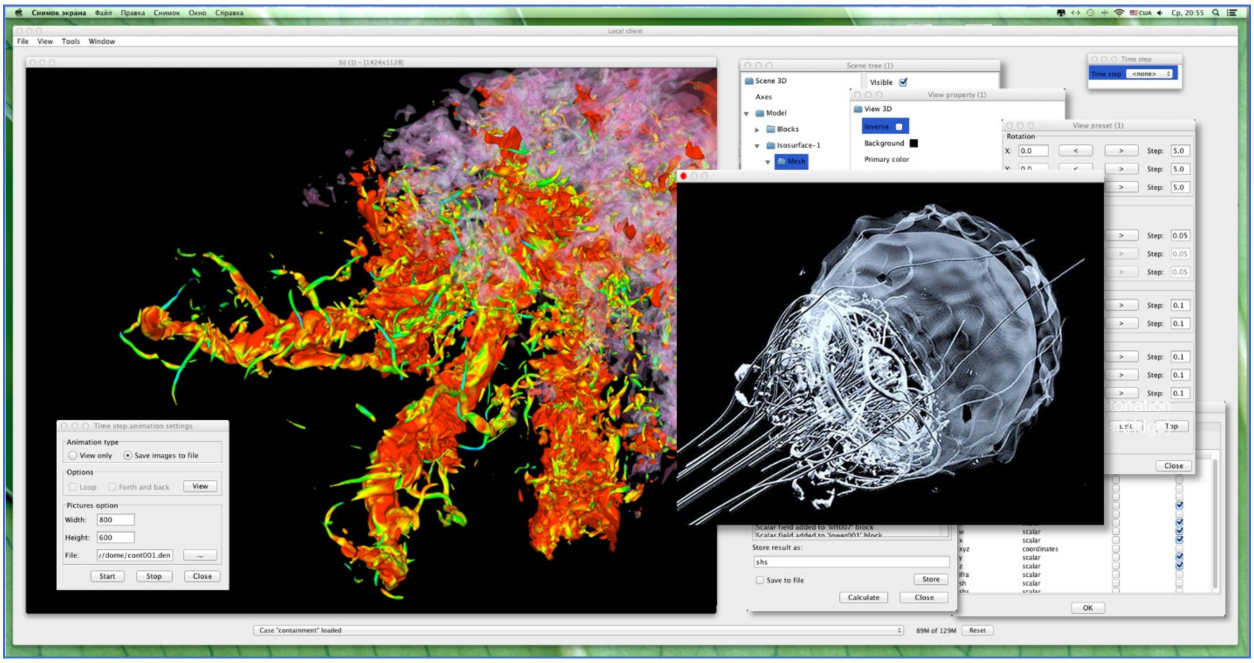

Figure 1 - GUI of system HDVIS

The addition of new functions in visualization system was carried out taking into account the experience of recent updates of the functionality of such systems as well as taking into account the methods that have become widespread in the processing of data for solving large problems of CFD in multiphysics formulation and the features of visual representation in computational astrophysics.

Interactive analysis in the visual system is based on dataset from solvers in a sequential or parallel processing mode. The results of mathematical modeling in computing systems using different spatial discretization are transferred for postprocessing with preliminary data structuring, declared in the calculation code. The obtained simulation results are stored in data arrays that repeat the block-structured curvilinear or regular meshes used in solvers, sometimes combined with AMR (Adaptive Mesh Refinement) patches that included in computational grids. High resolution patches are used to improve the quality of simulation and increase the accuracy of calculations in the allocated spatial volumes. In visualization system, combined meshes can improve the quality of detection of flow features in difficult-to-analyze situations. To obtain more informative saturation of visualization scenes, derivative functions and additional tools to calculate secondary scalar, vector and tensor fields are built into the visualization system code.

The program is equipped with the necessary set of functions for recovering information during the transformation of physical and derivative fields, highlighting their features in high-gradient and discontinuous zones of flows, near shock waves, and in other areas of flows, which are important for the analysis of super- and hypersonic flow regimes. Rendering of complex graphic objects in the postprocessor environment is doing with possible hardware acceleration and parallelization of some operations on workstations. This is very useful when processing large amounts of data with multiple intersections of geometry and abstract objects in scenes combined into animation sequences.

3. Practice of visualization of coherent turbulent structures observed in multiscale simulations

In a number of completed projects of unsteady flows studied morphologically changed significantly unpredictably and unevenly both in space and time. The feature extraction of coherent structures that form due to flow or plasma instabilities, such as Kelvin-Helmholtz, Rayleigh-Taylor, Rayleigh-Benard, Richtmyer-Meshkov instability and some others was in-demand task in visual analyzing. Qualitative analysis required special interpretation techniques for showing and highlighting the features of such flows.

To identify areas of instability and form connected structures, a built-in formula calculator was used to perform vector and tensor operations in primary scalar and vector fields when calculating field gradients, divergence, rotation, external derivatives and other quantities necessary to determine the topology of flows. In postprocessing user can choose ways of using vector field topology for interactive extracted feature-based visualization of flow simulation data. The experience of using visualization system options and applying various techniques for creating visualization scenes is illustrated by the examples of modeling flows of different scales in some projects where the system HDVIS was used by various investigators.

3.1. Turbulent flows created in hydraulic turbine

In CFD simulations, time-dependent flow is imaged in a way that effectively allows analysis of pronounced or hidden vortex structures. This remains a challenging imaging problem that requires considerable ingenuity in the use of techniques familiar from experimental studies. The basic technique is associated with flow tracing, often used to visually depict vortex flows in hydraulic devices with vortex formations inside pipes and cavities that arise when bodies flow around turbines. This approach is demonstrated by video snapshots (Figure 2 and 3) from modeling of turbulent flow in an elbow draft tube of an axial-flow Kaplan turbine [13]. Fluctuating turbulent stream was calculated on the base of the RANS approach. Simulation was performed with an in-house code SINF based on the second-order finite-volume spatial discretization using the cell-centered variable arrangement and body-fitted block-structured grids [14].

Draft tubes of Kaplan turbine are a diverging tubes fitted at the exit of runner of turbine and used to utilize the kinetic energy available at the exit of runner. Research on optimizing the flow in these devices is important for the process of establishing a mode of full energy recovery. The focus in study [13] was to check sensitivity of the predicted pressure recovery and outlet energy non-uniformity to wide variations in the inlet boundary conditions for transported turbulence quantities used in turbulence models, such as the standard k-ε model, the Wilcox k-ω model and the Menter SST model. The computations were performed for a draft tube with two outlet channels tested at an air test rig in combination with a runner. For comparison with experimental results, the applied visualization methods and calculation of integral characteristics were widely used. When analyzing the swirling flow inside the draft tubes, various methods were used to determine its structure and hydraulic characteristics that affect the energy loss of the flow. Tracking the movement of fluid particles with the generation of pathlines and pathstreams surfaces was an effective method of visualization.

The technique of combining the image of moving particles and the construction of Q-criterion isosurfaces with the choice of starting tracing from the grid nodes of the formation of these vortex surfaces allows you to analyze swirling flows in the most representative way (Figure 2 and 3). When constructing visualization scenes, we practiced calculation of translucent iso-surfaces for variables as a pressure, kinetics energy and velocity vectors components or derivatives fields, in combination with animation of video scene with surfaces of pathlines.

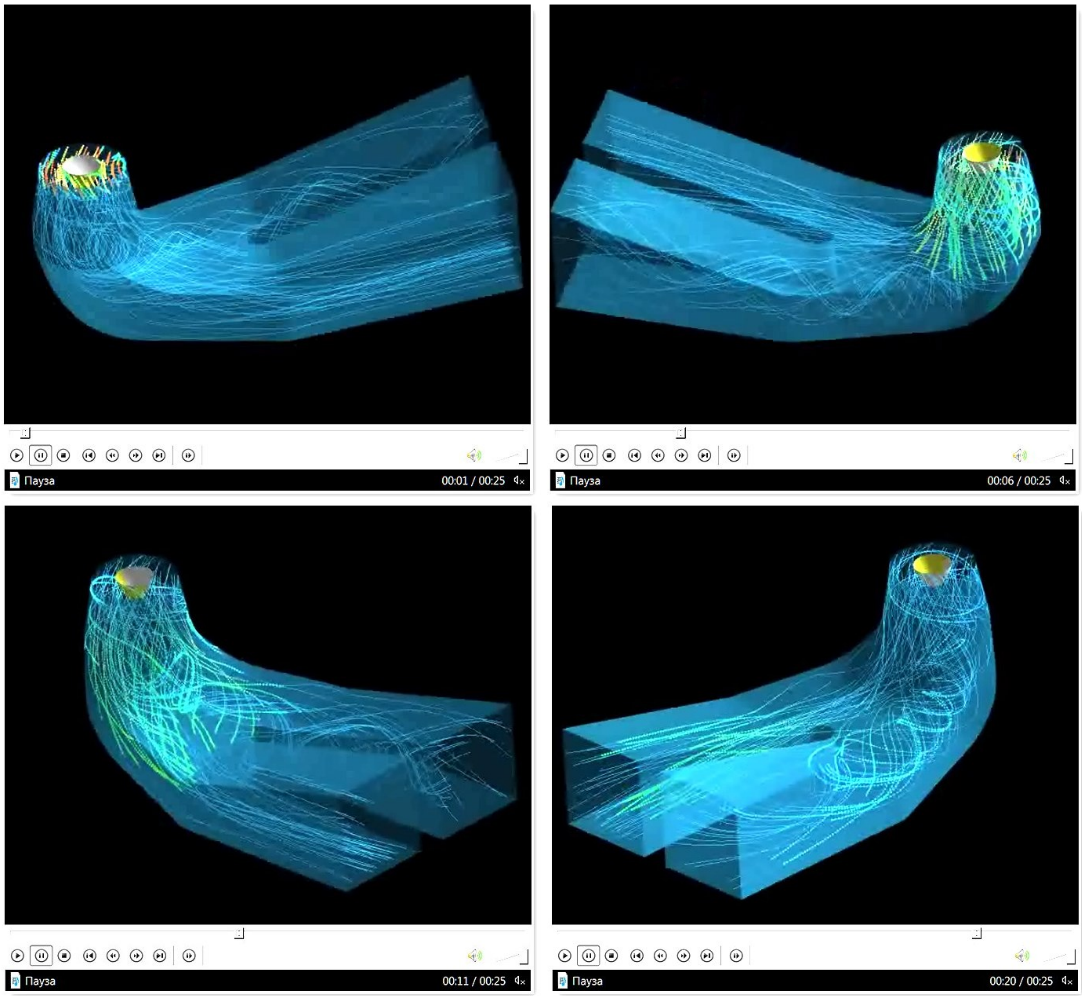

Figure 2 - Animation created from sequence of frame set of the spread of fluid particles that passing via draft tubes with a sharp turn

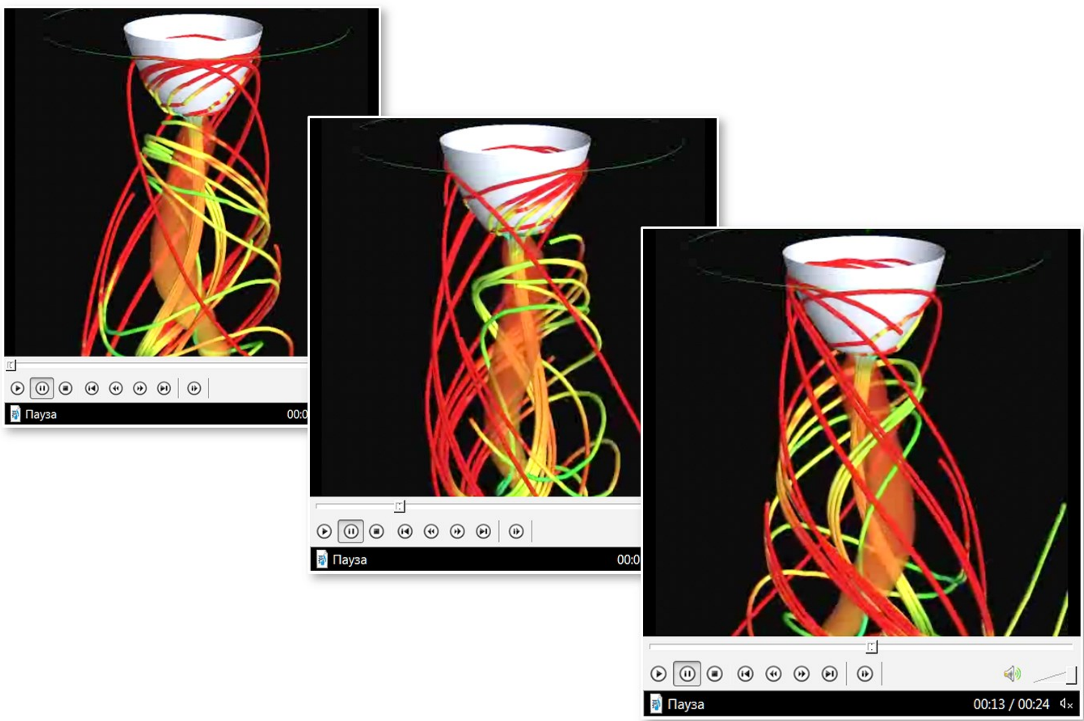

Figure 3 - The animation frames show spiraling streamlines around a precessing vortex rope

Using options of visualization system it is possible to set the points of passage of marked particles (including start and finish) in any reference points of generated geometric objects (isosurfaces of derivate abstract objects, spatial construction surfaces, lines of curvilinear coordinate system etc.). This makes it possible to efficiently calculate and choose the trajectories of moving particles, which ultimately leads to a better disclosure of the structures of vortex formations. In many cases, the idea of choosing intermediate positions points at the nodes of such geometric abstract objects as the tensor invariants, Q and/or □□ criterion surfaces turned out to be productive. Particle motion and pathlines animation techniques combined with integral calculations have been widely used to enhance and highlight vortex structures in visual analysis.

3.2. Supersonic flows in rotary detonation engine

When analyzing flows in power equipment, the most common and traditional visualization method is the construction of isosurfaces and rendering maps with distribution of various dynamically changing quantities. When modeling supersonic flows, the main techniques are combined to reveal high-gradient zones and layers. In many cases, meaningful visualization requires special techniques to enhance the image quality of video scenes.

Possible situations are illustrated with some results of modeling supersonic flows in promising energy devices. Over the past decade, significant progress has been made in the field of simulation a deflagration combustion mode of gases in rotating detonation engines (RDE). Combustion under pressure in high-speed ramjet and rocket engines has become the subject of interest in a large number of experimental and numerical studies, in particular in the serial studies noted in [15, 16]. The mathematical model describing the processes in the combustion chamber include balance equations for a multicomponent mixture of fuel and oxidizing agent, with chemical transformations and turbulent transfer of mass, momentum, and energy. Authors [16] have studied features of the implementation of combustion for different fuel and oxidizer supply (hydrogen/methane/acetylene - oxygen mixture) and different types of the combustible mixture: rich, lean, and stoichiometric. In a numerical experiment carried out several regimes of single-wave or multi-wave stable detonation, were studied for different mixture.

The sudden transition from a deflagration to a detonation type of explosion is stochastic. In calculation it is difficult to reliably determine streams in narrow flow zones with a high-gradient change in energy characteristics. When analyzing the calculations, this is required careful visualization and identification of the details of the combustion of the mixture at the detonation front in the inlet region near input channels orifices. A detailed three-dimensional numerical simulation of combustion inside the annular combustion chamber was carried out with visualization of fast processes during the transition from combustion to detonation using functionality of HDVIS [17, 18]. For informative visualization of subsonic and supersonic flow, combined methods of representation of semi-transparent isosurfaces, high relief and maps for different scalar and vector components were used, with different degrees of transparency and illumination of the desired zones shaded each other by walls of annular combustor with a central body and a cylindrical shell. This was combined with the display of contour lines of different values or accompanying them and interesting for analysis with their joint influence on each other. In the case of closed surfaces in combustion chamber, a translucent representation of multi-color maps and contours on the back and front sides of the curved section was often used. Special measures were required to isolate the image on the side closest to the observer. In graphical analysis, it was quite natural to set an adjustable mode of visual traversal of such surfaces. Such situations are often encountered in the analysis of gas flows in complex-shaped combustion chambers. The complexity of video analysis lies in the graphical representation of instantaneous detonation of combusting gas stream that is difficult to capture and illustrate intelligibly and expressively.

Some idea of combustion details in RDE using visual system are given by illustrations from [17, 18] (Figures 4 and 5). Shown here are some moments of propagation of wave shock front in the annular space of chamber from the place where fuel and oxidizer are supplied and to the exit of combusted fuel gaseous components. Figure 4 shows the colored contours of the three main combustion quantities: acetylene molar concentration, oxygen concentration and temperature on the surface of the internal cylindrical body, as well as the colored high relief of these quantities distributed above the micro-injector set.

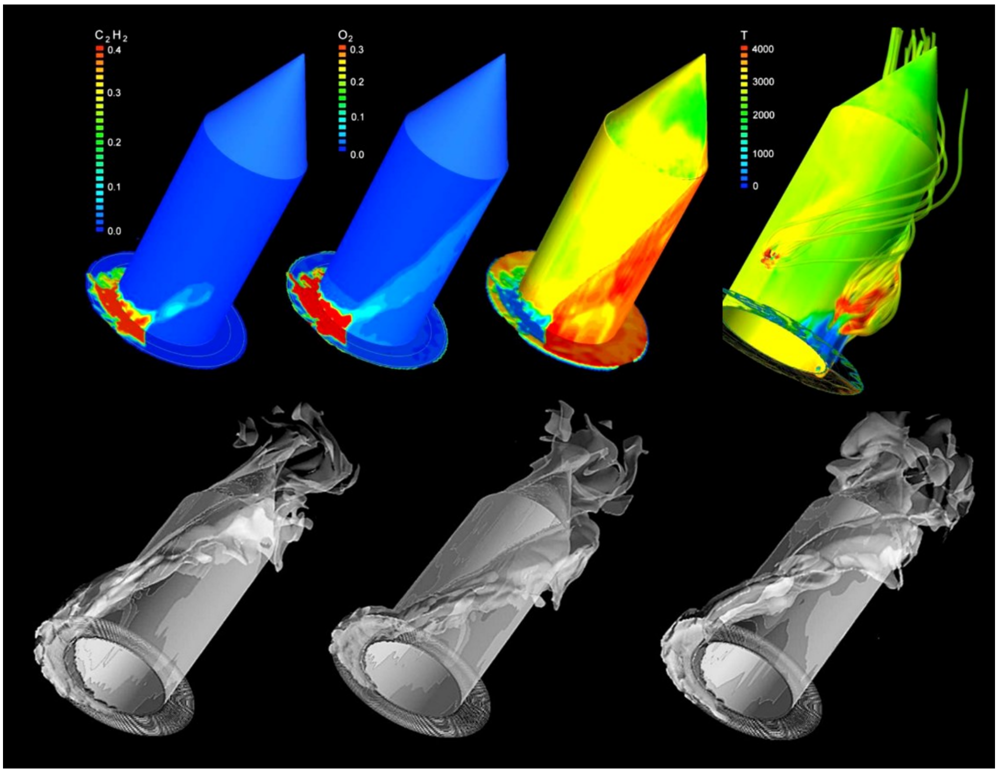

Figure 4 - Monitoring various characteristics of acetylene combustion in the detonation mode of operation of the RDE

The moment of development of a local vortex jet from highly heated near-wall spot near annuli central wall rotated around the axis is additionally shown for clarity. The spread of the vortex jet is shown by highlighting the streamlines from spot area, color reflecting the local gas temperature. In above views of transparent vortex veil imitated concentration field around the central body, a gray color palette is used. This approach helps to increase the clarity of images and the perception of the spatial arrangement of objects.

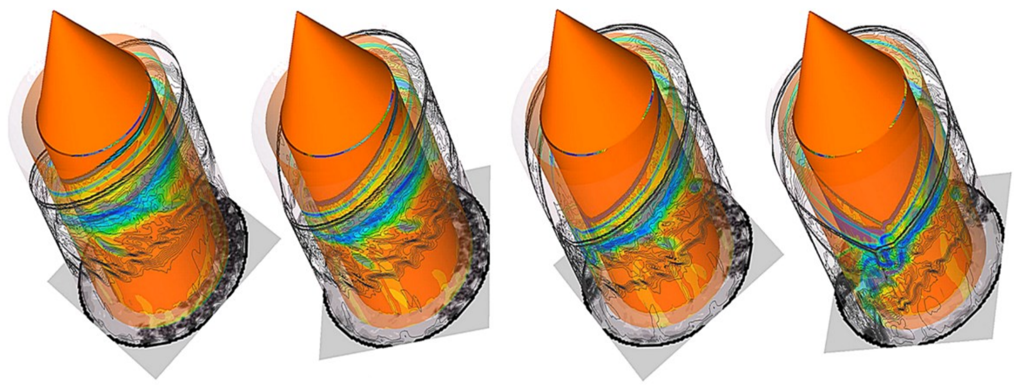

Figure 5 - Perspective bypass of the surveillance video camera around object used to show the details of the occurrence of detonation wave instability and its propagation in annular channel

On the animation frames of the propagation of the detonation wave, the outer wall of the combustion chamber is shown as transparent. It shows schlierens of gradient wave front of the combustible mixture and traces of the wave-like development of Richtmyer-Meshkov (RMI) and/or possible Kelvin-Helmholtz instability (KHI). The user of video system can trace the temporal changes in the combustion process with viewing the surface deformation of the detonation shock wave and the line of oblique shock wave passage along the channel solid surfaces until the combustion products exit the chamber. A slip line is displayed between freshly detonated products and older products. It is possible to trace the development of the secondary shock wave, view the mixing areas between the fresh pre-mixed fuel-oxidant gases and the detonated gases. This can be seen in the respective flow details in the area with blocked micro-injectors and in the neighboring zone of the unreacted premixed fuel-oxidant mixture. One can trace the connections between combustion modes and much more processes through visualization tools used. Visual studies have help more clear understanding results of gasdynamics modeling in different combustion modes in chamber and to add description of different vortex effects that will induce pressure oscillations at the RDE nozzle exhaust. The practice of processing simulation data of the perspective engine under study showed the need to create set animations of gasdynamics processes in the combustion chamber and more convenient control of the HDVIS options in corresponding video processing modes.

3.3. Coherent structures of transient flows

When conducting a visual analysis of the results of simulation of flows with different spatial scales and different speed ranges, feature extraction methods are often used to identify eddies in coherent structures using tensor invariants, velocity components, and their gradients. The calculation of various types of derivative fields in the discussed visualization system was carried out using the built-in formula calculator. The most frequently used operation was the detection of vortex structures with the calculation of the Q and λ_2-criterions [19] with their definition in terms of the instantaneous velocity gradient tensor [20]. The Q-criterion is applied to transitional and turbulent flows where the interaction of coherent vortices is found to produce large-scale Λ-shaped and hairpin-like vortices. This technique of visualization was used in treatment of results in direct numerical simulation of instabilities flow transition to turbulence in a supersonic boundary layer on a flat plate [21]. The modeling was based on a direct numerical solution of 3D unsteady Navier–Stokes equations using in-house program package CFS3D [22]. Numerical simulation revealed the mechanism of boundary layer development, which includes the stages of linear growth of perturbations, stochastization, and formation of a turbulent flow. The calculation results showed that the preliminary laminar flow in the inlet section was excited by the most rapidly growing disturbances represented the three dimensional Tollmien–Schlichting unstable waves, which arises in a shear flow propagating at the angle to the main flow direction.

An example of revealing the structure of the transient flow regime for one time moment is shown in the Figure 6. The figure shows how a laminar-turbulent flow transition is formed, passing through the stages of development: inflow with the formation of a pair of counter-rotating filaments; the formation of primary rolls and strips; tortuous and varicose discontinuities; the manifestations of KH instability with Λ-shaped and hairpin vortices. In a series of animations, it was demonstrated how flow going through stages of transition, and how instability leads to stochastic perturbations of flow and local pulsations lead to folding of its into filaments of concentrated vortices. The Λ-shaped vortex becomes the hairpin vortex by self-deformation as this shown in figure below. The video scene based on a combination of displaying translucent Q-isosurfaces colored with distance from plate, and white colored particles moving along and across loop vortex trajectories. Animation of vortices allocated by the Q criterion showed that the observed formations completely repeat the horseshoe vortex model of wall-bounded turbulence, noted by T. Theodorsen [23]. Λ-shaped vortexes morphologically poorly distinguishable from hairpin vortices (shown in the corner fragment). The negative image in magnifying glass is used to highlight the detail image of the mushroom-like volumetric vortex fragment.

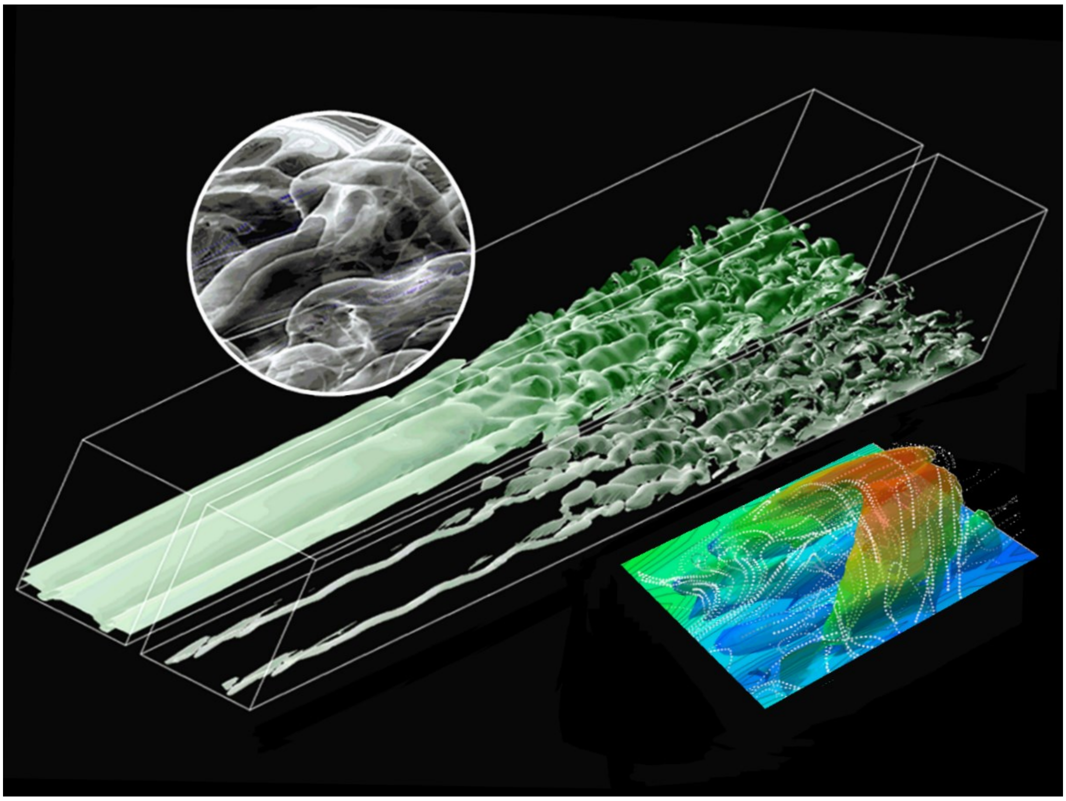

Figure 6 - Vortex structure of supersonic flow during the transition from laminar to turbulent regime with the selection of mushroom-shaped and hairpin-like vortices

A series of other studies examined the situation with a different form of gas movement during convective vertical ascent of air. The development of air near-wall turbulence during the ascending convective movement of the flow has a similar character, but differs in its “falling” structure when gravity is applied to the emerging vortices and local wall heating conditions. These features have been noticed when visually analyzing the results of direct numerical simulation of natural air convection near a vertical isothermal hot plate in [24]. Formation of vortex flow structures in this situation is illustrated in Figure 7.

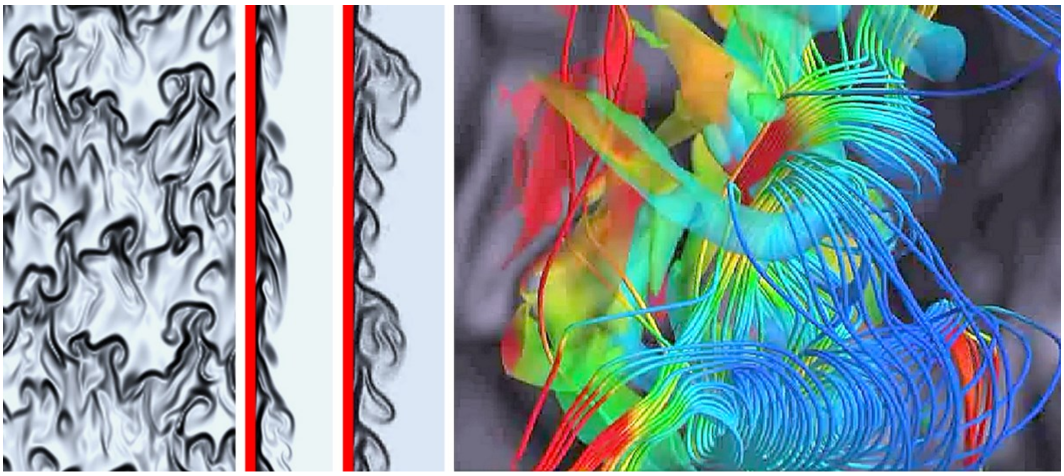

Figure 7 - Hairpin vortices shown by Q - criterion surfaces that colored by local velocity value and shadowgraph of vortex cascade near heated vertical plate

On the left part of picture a numerical shadow image confirms the structure of the heated air flow in plan and cross projections, showing the observed vortex redistribution in flow investigated. Used technique was supplemented by the construction of numerical interferograms of the boundary layer, imitated in visualization system. It can be seen that the bending of the vortex heads with their rotation towards the base of the vertical wall and departure of unstable vortex filaments from the wall towards the colder zone of the air flow. Packets of hairpin vortices are aligning in streamwise generate new loop-shaped vortices downstream of its own position. The vortices cause intense oscillatory motion of the flow and are responsible for the transfer of cold air from the outer region to the hot wall and the release of high-temperature portions of air from the wall surface into the surrounding layers of air. Oscillatory redistributions of the wall heat flux over the wall surface indicate clearly that the vortices have a strong effect on local heat transfer in the region behind the legs of the hairpin-like vortex. During the evolution of convective flow movement, the concentrated vortices amplify and take a hairpin-like shape. It is remarkable that the head of the 'hairpin' is inclined opposite to the main flow direction in contrast to the case of the forced flow over horizontal plate where the hairpin legs are directed opposite to stream. It is remarkable that in the natural-convection flow, the 'legs' of the vortex are located in the region of higher-speed flow. Combined imaging techniques used have made a contribution to establishing the nature of the resulting vortex effect.

3.4. Visualization technics in astrophysics simulation

The possibilities of visualization in the HDVIS environment were widely used in solution and analysis of some problems in numerical astrophysics. For a number of years, the consequences of shock interaction during the passage of a shock wave generated by a supernova explosion with the matter of molecular clouds have been studied and studies have been carried out on the consequences of collisions of molecular clouds with each other using various scenarios of impact and disintegration of the resulting structures.

For example according to the problem set in work [25], initially spherical clouds interacted with the post-shock medium of supernova blast remnants. Gas flow evolution of molecular clouds was derived by solving the 3D Euler equations of mass, momentum, and energy conservation.

High resolution numerical grids with more than four billion nodes were used in parallel calculations on multiprocessor hybrid computers. It required visual processing and analysis of results obtained over a long period of time, with significant volumes of output data at the terabyte level. The capabilities of the presented visualization system when working with such large volumes of calculation results were used as widely as possible. Visual analysis of dynamic changes in molecular clouds with a complex vortex structure using a graphical representation of the Q-criterion can be used, often in combination with other characteristics, for example - denstrophy, which is an indicator of compressible turbulent velocity fluctuation (Figure 8).

One can note that the technique of volumetric visualization with opacity of vortex-like objects often produces images that are difficult to perceive. In the case of using translucent surfaces, the picture of vortex formations also becomes confusing and difficult to perceive due to their huge number and mutual shading. In this situation, the visual analysis can be improved by using the animation features available in the system. The method of obtaining time-sequential numerical schlieren images in different sections with the presence of high-gradient mixing zones of cloud matter with explicit highlighting of instability to be convenient from an informative point of view. A good dynamic illustration of the processes was the animation of changes in the density gradient structure of matter and morphological changes of clouds during deformation over time after the passage of a shock wave. The sample of such animation of vortex generation inside of clouds is shown in Figure 9. In meridional section of gas matter of current formation, one can see a vortex structure stimulated by Kelvin-Helmholtz instability at the outer and inner clouds boundaries, and vortex consequences of Richtmyer-Meshkov instability behind the shock wave front. Double and triple separation points of oblique shock waves and Mach compressed legs are clearly distinguishable in the gradient zones of the synthesized wave structure and are fully identified in the movie with the numerical schlieren sequence.

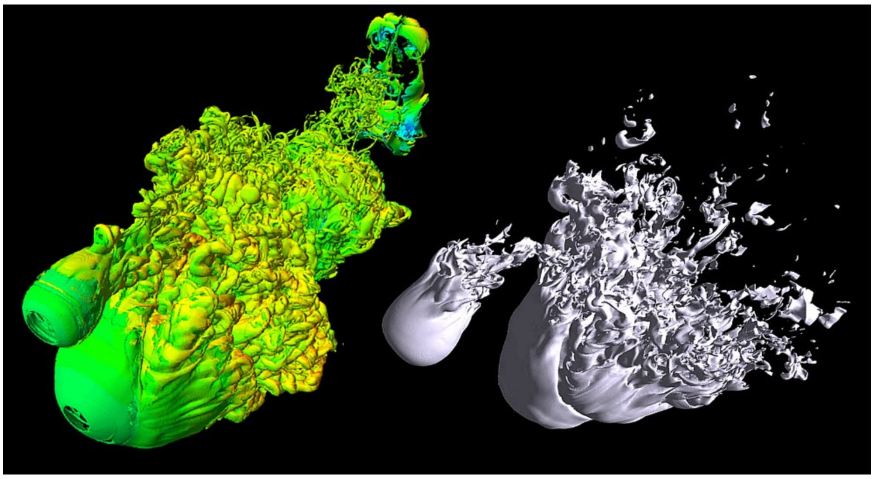

Figure 8 – Visualization of spatial vortex formation of molecular clouds after interaction with a shock wave from a supernova explosion using denstrophy and Q-criterion indicators

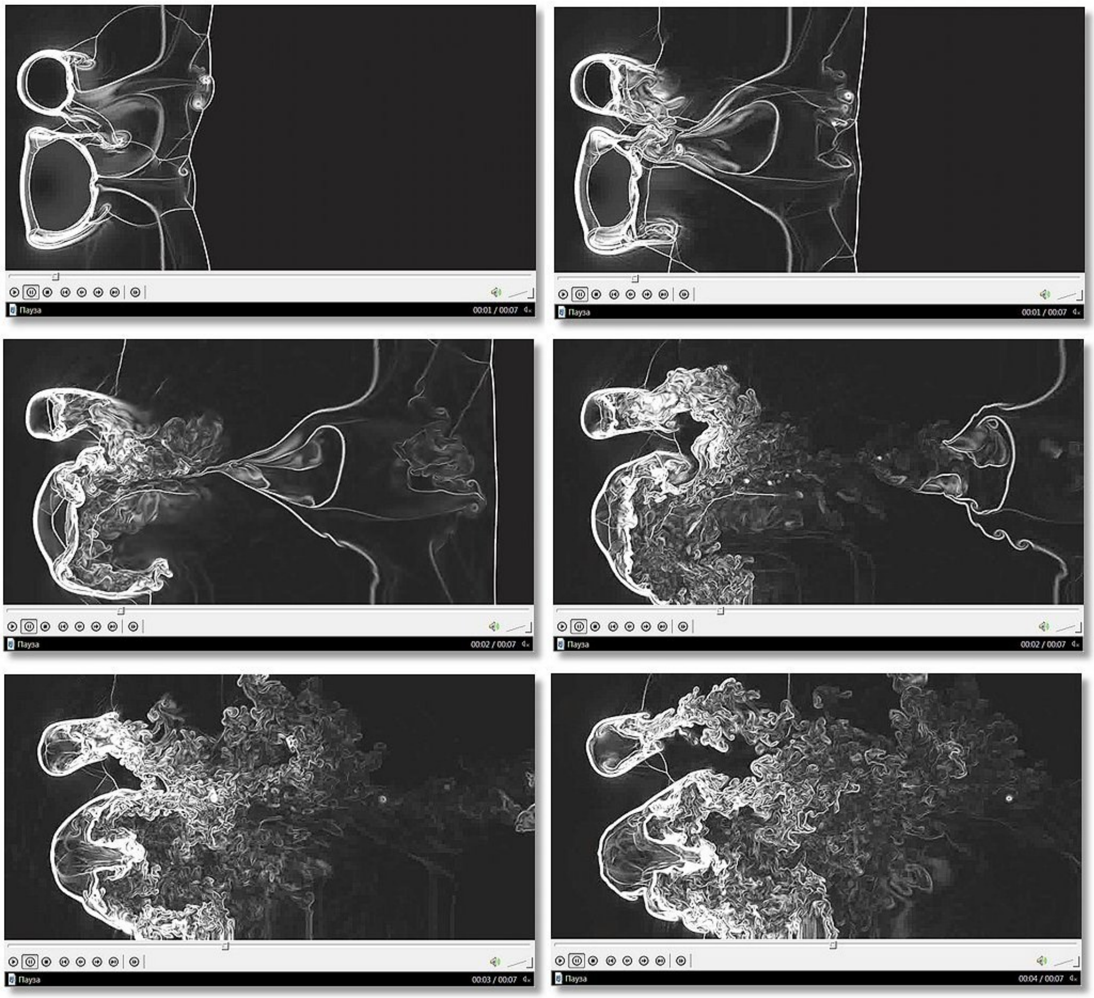

Figure 9 - Vortex structure of molecular clouds mixed after a collision with a shock wave generated a supernova explosion with vortices highlighted in the numerical schlieren

In a numerical experiment completed with various scenarios of interaction of impact molecular clouds, a set of calculations was carried out on possible schemes for the collision of giant molecular clouds: with direct and displaced impact, taking into account the rotation of clouds and without it, identifying possible dense zones of nebula in which gas and dust are contracting, resulting in the formation of new stars [26]. The peculiarities of molecular clouds and clumps, residuals and gas shells with fragmentation of filamentous structures were analyzed using a combination of different tool in video system under discussion. It has been established that during a collision in the stagnant zone of clouds, hypersonic turbulence arises and intensifies with the formation of filamentary structures, significant stratification of gaseous matter, and disintegration of clouds with intense vortex transformation of the emerging structures. Taking into account the rotation of nebulae with molecular clouds around a common axis made it possible to reveal additional effects of the growth of fragmentation of clouds remnants and the formation of clumps with a sharp increase in the density of matter inside them during collisions. A picture of the consequences of collision of contrary rotated molecular clouds with compacted clumps and spiral-like filaments is shown in Figure 10.

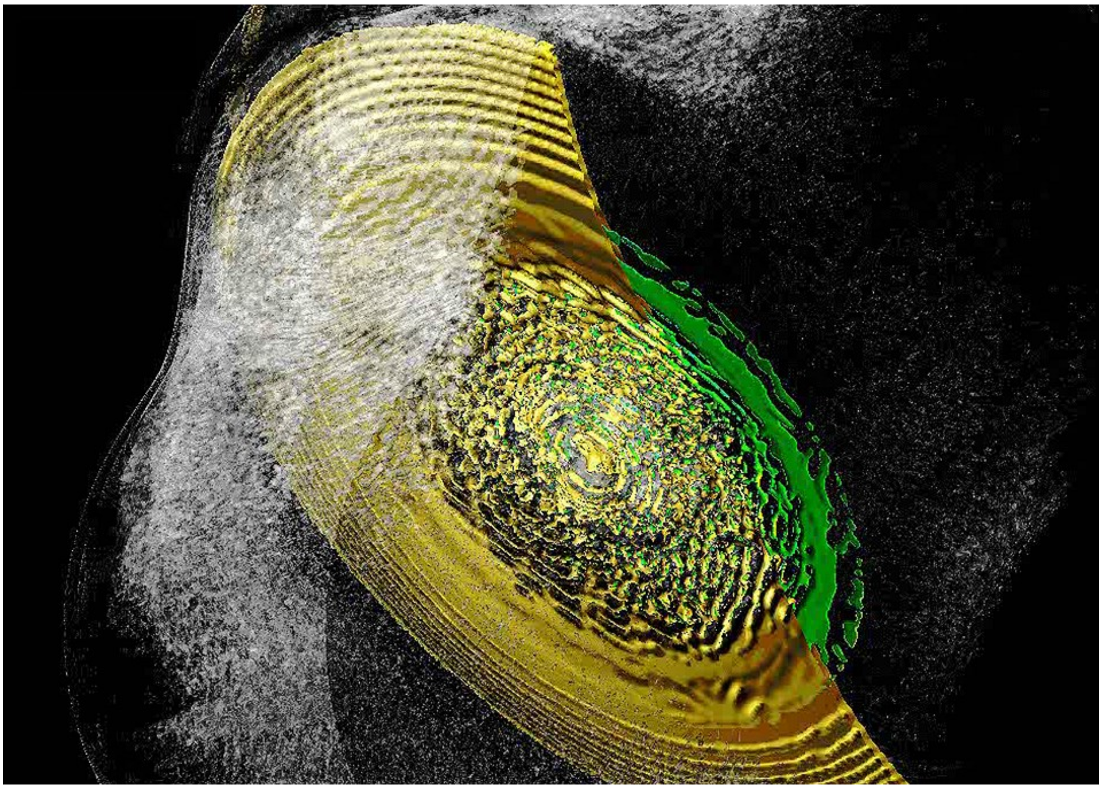

Figure 10 - Clumps and gas filaments shown in combine with condensed gas sink zones in which gravitational collapse can be initiated during collision of molecular clouds

Rotation in clouds with spiral transfer before stagnation zone leads to radial redistribution of compressed gas in the collision core and accelerates flexural corrugation of core before destruction. Modeling has shown that ring wave disturbances of matter in a compressed core can be caused by the Kelvin-Helmholtz instability, and non-linear thin-shell instability, leading to local density disturbances in formed clumps and filaments. The influence of variable regimes of colliding clouds on the change in the shape of the lenticular compression zones in the region of the main impact was analyzed via serial animation of cosmic scale time process. Long-term processing of serial sequences has been sufficiently optimized when working in presented visualization environment. Simulation of rotated colliding molecular clouds made it possible to clarify the details of the origin of turbulization and shape morphing of structures inside molecular cloud remnants, in clumps and filaments. The simulation of the giant molecular clouds collision revealed the conditions for reaching the critical density in fragmented clumps corresponding to the prestellar consolidation of internal stellar medium matter.

4. Conclusion

High-resolution and detailed visualization of multi-scale unsteady transitional and turbulent flows in various energy devices, engineering structures, supersonic and hypersonic flows of various natures is used in analyzing the results of solving large problems in computational fluid dynamics and numerical astrophysics. In the applied computer programs implemented on supercomputers, it was possible to parallelize calculations using various approaches. In the developed HDVIS system, many rendering operations are parallelized too, which made it possible to effectively carry out analysis on graphics stations with a sufficiently large RAM. Presented authorized visualization system is used to process to treatment of output results in relation to the developed new computational codes. Advanced system has proven to be effective in analyzing the results of huge-scale simulations for solving many flow modeling problems of a wide variety of scales. The experience of using visualization system options and applying various techniques for creating visualization scenes are illustrated by the examples of modeling flows in different projects where the system was used by various investigators.

5. Acknowledgements

The reported study was supported by Russian Foundation for Basic Research according to the research projects 17-07-00569 and 19-29-09070.

References

1. Visualization Toolkit (VTK): https://vtk.org.
2. Bondarev A. E., Galaktionov V. A.: Current visualization trends in CFD problems, Applied Mathematical Sciences, Vol. 8, No. 28, 1357-1368 (2014).
3. Bondarev A.E., Galaktionov V.A., Chechetkin V.M.: Analysis of the development concepts and methods of visual data representation in computational physics. Comput. Math. and Math. Phys. Vol. 51, 624–636 (2011).
4. Volkov K. N., Emelyanov V. N., Teterina I. V., Yakovchuk M. S.: Visualization of vortical flows in computational fluid dynamics, Computational Mathematics and Mathematical Physics, Vol. 57, No. 8, 1360–1375 (2017).
5. Iakobovski M., Nesterov I., Krinov P.: Large distributed datasets visualization software, progress and opportunities, Internet-Journal “Computer Graphics & Geometry”, Vol. 9, Issue 2, 1-19 (2007).
6. Lan F., Young M., Anderson L., Ynnerman A., Bock A., Borkin M.A., Forbes A. G., Kollmeier J. A., Wang Bei.: Visualization in Astrophysics: Developing New Methods, Discovering Our Universe, and Educating the Earth, Computer Graphics forum, Vol.40, Issue 3, 635-663 (2021).
7. https://developer.nvidia.com/nvidia-index.
8. Schneider E. E., Robertson B. E.: CHOLLA: a new massively parallel hydrodynamics code for astrophysical simulation, The Astrophysical Journal. Vol. 217, Issue 24, 1-34 (2015).
9. Goryachev V.D., Balashov M.E.: Visualization of simulation results in Information and Computational System, Proc. of Int. Conference “NIT&QM'2009”, Ed.: Tikhonov A.N. and al., SSI FSU ITT “Informica”, 29-31 (2009).
10. https://wci.llnl.gov/simulation/computer-codes/visit.
11. https://www.tecplot.com/products/tecplot-360.
12. Haber R.B. and McNabb D.A.: Visualization idioms: A conceptual model for scientific visualization systems, Visualization in Scientific Computing, 74–93 (1990).
13. Korsakov A.B., Smirnov E.M., Goryachev V.D.: CFD modeling for performance predictions of a hydraulic turbine draft tube: the effect of inlet boundary conditions for two-

equation turbulence models, in: Proc. 5th Int. Conference on Modeling Fluid Flow (CMFF'12), BUTE, Hungary, 757-763 (2012).

14. Smirnov E.M., Zajtsev D.K., The finite-volume method in application to complex-geometry fluid dynamics and heat transfer problems, Scientific-Technical Bulletin of the St.-Petersburg State Technical University, Vol. 36, Issue 2, 70-81 (2004).

15. Xie Q., Ji Z., Wen H., Ren Z., Wolanski P., Wang B.: Review on the Rotating Detonation Engine and its Typical Problems, Transactions on Aerospace Research, Vol. 2020, Issue 4, 107-163 (2020).

16. Mikhalchenko E.V., Nikitin V.F., Phylippov Yu.G., Stamov L.I.: Numerical study of rotating detonation onset in engines, Shock Waves, Vol. 31, Issue 7, 763–776 (2021).

17. Rybakin B.P., Goryachev V.D., Mikhalchenko E.V., Nikitin V.F., Stamov L.I.: Visualization of processes in an engine with a rotating detonation wave, AIP Conference Proceedings, Vol. 2304, Issue 1, 1-5 (2020), DOI: 10.1063/5.0034707.

18. Mikhalchenko E.V., Nikitin V. F., Goryachev V.D.: Simulation of the Operation of a Detonation Engine, In: Indeitsev D.A., Krivtsov A.M. (eds.) Advanced Problem in Mechanics II, APM 2020. Lecture Notes in Mechanical Engineering, Springer, 98–107 (2022), https://doi.org/10.1007/978-3-030-92144-6_7.

19. Hunt J.C.R., Wray A.A., Moin P.: Eddies, streams, and convergence zones in turbulent flows, Centers for Turbulence Research Report CTR-S88, 193-208 (1988).

20. Chong M.S., Perry A.E., Cantwell B.J.: A general classification of three-dimensional flow fields, Physics of Fluids, Vol. 2, 765-777 (1990).

21. Khotyanovsky D., Kudryavtsev A., Goryachev V.: Parallel DNS of Transition and Breakdown to Turbulence in a Supersonic Flow over the Flat Plate, 23rd Int. Conf. Parallel CFD2011, Book of Abstracts, Barcelona Supercomputing Center, 39-40 (2011).

22. Khotyanovsky D.V., Kudryavtsev A.N.: Direct numerical simulation of the transition to turbulence in a supersonic boundary layer on smooth and rough surfaces, Journal of Applied Mechanics and Technical Physics, Vol. 58, 826–836 (2017), DOI: 10.1134/S002189441705008X.

23. Theodorsen T.: Mechanism of turbulence, Proc. Midwest. Conf. Fluid Mech., 1–18 (1952).

24. Abramov A.G., Smirnov E.M., Goryachev V.D.: Temporal direct numerical simulation of transitional natural-convection boundary layer under conditions of considerable external turbulence effects, Fluid Dynamics Research, Vol. 46, Issue 4, 1-17 (2014), DOI: 10.1088/0169-5983/46/4/041408.

25. Rybakin B., Goryachev V.: Modeling of density stratification and filamentous structure formation in molecular clouds after shock wave collision, Computer & Fluids, Vol. 173, 189–194 (2018), DOI: 10.1016/j.compfluid.2018.03.009.

26. Rybakin B., Goryachev V.: Simulation of Molecular Cloud Collision Dynamics Using Heterogeneous Systems, Journal of Applied and Industrial Mathematics, Vol. 17, No. 1, 168–175 (2023), DOI:10.1134/s1990478923010192.