Momentum Waves, Vol. I, No. 1, 1-29, 2019 (06.12.19)


# Searching for Waves in the Incompressible Navier-Stokes Equations - The Adventure

Desmond Aubery[1]*

**Abstract**
This article traces a journey of discovery undertaken to search for wave phenomena in the incompressible Navier-Stokes equations. From the early days of my interest in Computational Fluid Dynamics (CFD) used for consulting purposes, and the use of various commercial solvers eventually leading to research programs, at a number of universities, spanning a number of years. It reviews research programs at Dortmund University (Dortmund, Germany), and this author's post-graduate study at Chulalongkorn University (Bangkok, Thailand). During the latter, it was noticed that flow solutions became unstable when certain combinations of parameters were used - especially when real-life density and viscosity were used. Clarity was needed. This author had a perception that this research could lead to an understanding of the turbulence phenomenon, Tensor calculus was used to understand the macro nature of the NS Equations, and to place them firmly into the family of wave equations. My research continued in private for some 15 years, with the occasional presentation of findings at conferences, and an internet blog. In recent months major breakthroughs have been made, and now the evidence for wave phenomena is convincingly demonstrated.

**Keywords**
Navier-Stokes — Incompressible — Waves — Momentum Waves

[1] *Rarefied Technologies (SE Asia)*
***Corresponding author**: auberydes@gmail.com

## Contents



## Introduction

This article traces a journey of discovery undertaken to search for wave phenomena in the incompressible Navier-Stokes equations.

My interest in Computational Fluid Dynamics (CFD) began during my years as an international consulting engineer, where information about liquid and gaseous flow regimes was needed. A number of commercial solvers were used at that stage. This interest eventually lead to research programs, at a number of universities, spanning a number of years. Prof. Stefan Turek and his *Featflow/Feast* team at Dortmund University (Dortmund, Germany), were very kind, in hosting me a number of times, allowing me access to their computers, and answering many of my questions.

Still more questions than answers were found, however, and this inspired a personal research program that began during my post-graduate studies at Chulalongkorn University (Bangkok, Thailand). During two courses, one *Advanced computational fluid mechanics (FVM)*, and the other *Finite Element Method for CFD (FEM)*, it was noticed that flow solutions became unstable when certain combinations of parameters were used - especially when real-life density and viscosity were encountered. CFD had a history of solution divergence. Why? Many 'steady state' solutions either diverged, or provided rather non-physical behavior. More clarity was required - an underlying fundamental mechanism appeared to be at work. This author had a perception that this research could lead to an understanding of the turbulence phenomenon.

How could numerical answers be trusted if we were not able to explain basic phenomena?

We know a great deal about low-speed, laminar, viscous flows, and high-speed, inviscid flows. What do we really know about the flow regimes between those two extremes? Fluid mechanics explains the laminar flow regime, and the fully-turbulent flow regime. What happens in the *transition zone* for fluid flows? What occurs there? Why does the form of the flow change so dramatically? This question has interested scientists for a long time.

Tensor calculus and continuum mechanics became essential allies in order to understand the macro nature of the NS Equations, and to place them firmly into the family of wave equations. The research program continued in private for some 15 years, on a shoe-string budget, with the occasional presentation of findings at conferences, and an internet blog. In recent months major breakthroughs have been made, and now the evidence for wave phenomena is presented openly.

One of the distinct challenges of this project has been its cross-disciplinary nature. Mechanical engineering, physics, mathematics, computer science, information technology, visualization science, big data.

# 1. Research Overview

My exposure to Computational Fluid Dynamics (CFD) began in 1997, at a commercial research facility in Rugby, UK, where a new workstation and software package had been installed. This workstation was being used for automotive radiator fin optimization.

The first impressions were of flow instability over a 2D cylinder. Over a period of time, extremely complex and ambitious models were developed, including complex geometries. The models provided valuable insights into the heat-transfer phenomena occurring within the extremely confined corrugated multi-louver radiator fins. At that stage heat-transfer and pressure-drop accuracies were rather poor ($\pm 30\% - 50\%$). Comparative experimentally-derived techniques predicted heat-transfer $\pm 3\% - 4\%$ and pressure-drop $\pm 5\% - 8\%$.

## 1.1 Consulting / Commercial software

My passion for Computational Fluid Dynamics (CFD) began in 1999, during a period when the author was actively involved in international consulting related to the development and commercialization of an new copper-brass brazing process for automotive heat-transfer devices (*Cu*probraze®). It became necessary to understand the Nitrogen gas dynamics in Controlled Atmosphere Brazing (CAB) furnaces used to braze high-performance aluminum and copper-brass components.

At that time, the license for a commercial multi-physics modeling and solver package called *Algor* was purchased. This allowed the fluid flow to be modeled, and a short movie was developed. The fluid jet was seen to strike the lower portion of the chamber and remained very close to the lower wall. This provided deep insights into the gas flow dynamics.

The next commercial solver investigated was a package from CD-Adapco - *Star-CD*. This provided additional tools and insights. More capability was required, however. At the time, its license cost was prohibitive.

Another commercial package *Flo++* which was almost completely functionally compatible with *Star-CD* was then used. The learning-curve for this was, however, very steep, and its graphics capabilities presented challenges. Complex meshes had to be constructed using an external package and imported into *Flo++*. The graphics output were, however, excellent. Even with these challenges, a 25-fin multi-louver automotive radiator fin was modeled in a virtual wind-tunnel. The results were fascinating and very informative.

The single biggest drawback to the project, at that stage however, was the cost of CFD software. It was time to begin investigating Open Source software.

## 1.2 Feat Flow, Feast - Dortmund, University

In around 2000, Prof. Stefan Turek and his *Featflow/Feast* team at Dortmund University were acknowledged as world leaders in low-speed, incompressible FEM (Finite Element Method) fluid solvers.

*Featflow* was available for free download and independent researchers were encouraged to experiment. The first hurdle was learning *Suse* Linux. This opened up the world of Unix and mini/mainframe computers. A model could be simulated on a laptop computer, then transferred to run at high resolution on the main/mainframe systems.

The team were incredibly kind and hosted the author for a number of visits to their facility. To be able to use mini and mainframe computers was an amazing experience. They very patiently answered my numerous questions. At one point, the author presented a review of his research, and vision of one day being able to simulate and visualize a full-scale automotive radiator. (This is only now close to becoming a reality but would require enormous computational resources).

The single largest hurdle to using *Featflow* was generating, meshing, and converting to suitable *Featflow* format, the object to be simulated. The modeling and graphics tools, at that stage, were still in their infancy. Setting up boundary conditions was also somewhat of a challenge. For large, complex geometries, these challenges made the conversion from commercial CFD packages to the emerging Open Source software difficult.

As an academic research tool however, *Featflow* was superb.

The main visualization software package used at that stage was *GMV*[1]. It provided very useful, high-resolution graphics results. The human eye is able to capture minute changes when very high resolution graphics are displayed.

[1] http://generalmeshviewer.com

### 1.3 Chulalongkorn University, Bangkok

It was time to begin a more detailed research program.

This phase began as the study of the heat-transfer and flow patterns in automotive corrugated multi-louver radiator fins. During the program a number of courses were undertaken. During two such courses, as mentioned earlier, one *Advanced computational fluid mechanics (FVM)*, and the other *Finite Element Method for CFD (FEM)*, it was noticed that flow solutions became unstable when certain combinations of parameters were encountered - especially when real-life density and viscosity were used.

For both approaches, similar types of instability emerged. Solution divergence was often the norm. Time-step selection, in many cases, was critical to success, or failure, for a simulation. A simulation could run perfectly for a few hours, then rapidly diverge. What was actually happening in the flow during solution? No clear answer for the phenomenon was provided in the literature surrounding these phenomena.

This encouraged the author to begin monitoring solver output at each time interval. These temporal snapshots were then combined into movies. This allowed an understanding of the solution difficulties to emerge.

The following courses were taken during this research phase:

- Measurement and instrumentation
- Advanced finite element method (FEM)
- Finite element method for CFD (FEM)
- Advanced dynamics
- Advanced computational fluid mechanics (FVM)
- Convection heat transfer
- Radiation heat transfer
- Continuum mechanics

During this time, a number of research papers were written. Refer to the Gallery section for more detail.

### 1.4 Internet Discussion

During the Chulalongkorn study program, an interesting open discussion was entered into on the web forum *CFD Online*, which, at that time, consisted of a number of CFD specialists, mathematicians, fluid-flow researchers, as well as other interested academics.

These were the early days of CFD, when researchers still believed that CFD would provide the tools needed to explain the turbulence phenomenon. (Not long after this, ad hoc turbulence models and wave-restriction techniques became the order of the day).

During the discussions, a strong debate emerged between the author and an anonymous academic with a solid understanding of the mathematics of fluid flow, and incompressible flows - *Tom*. He pointed the author towards a number of extremely useful resources. The discussion initially centered around the topic - *"Can Shock Waves occur in incompressible Navier-Stokes fluid flows"*?

This open discussion went on for a few months. In the end, it became apparent that longitudinal wave phenomena could not emerge in incompressible Navier-Stokes fluid flows. Thus, compressive shock waves as seen in high-speed flows, were not possible in incompressible flows. This was later confirmed in the works of Truesdell [Truesdell and Toupin, 1960] [Truesdell and Rajagopal, 2000] [Truesdell and Noll, 2004].

However, based on Truesdell, the possibility emerged that other types of wave phenomena and singular surfaces could exist within a flow space, and not violate the incompressible constraint. This fact became clear as the research project progressed. This project owes a great deal to *Tom* and Truesdell.

### 1.5 Self-sponsored Research

During 2006/7, as findings were emerging, the external resistance became almost insurmountable. There were no books or papers to read on the subject. It appeared to be entirely new ground. Books about wave phenomena, in general, existed, but none relating to incompressible fluids.

The project's research findings appeared, at that time, to be controversial to many academic colleagues. The findings had the potential to explain the *turbulence* phenomenon. This had the potential to challenge the long-standing perception of turbulence being a chaotic phenomenon, not deterministic, and was viewed with skepticism.

Practically, no-one wanted to shoulder the risk to their academic reputation by backing an outside upstart with extremely challenging academic views. At the time, it was incredibly demotivating. In hindsight, however, their position was entirely understandable.

The author was advised by a colleague of long-standing to research and publish alone. From 2008 it was felt that the project would be better served if all research was conducted privately, and self-sponsored.

Thus during this phase extensive reading and research was undertaken in regards to wave phenomena. A private library was built of almost every wave-related and high-level mathematics book that could be purchased at that time.

### 1.6 Waves and Structures Blog

In 2015 a blog was developed to open up some of the research findings and to encourage collaboration - *"Navier Stokes Waves & Structures"*[2]. It had a reasonably solid global following for a few years. Researchers were free to download papers and pictures. Sadly, no-one decided to communicate, acknowledge or collaborate. It was very lonely at times.

To all colleagues who downloaded papers and pictures, I hope that the information offered you new insights into this new wave paradigm. For others, confusion may have reigned.

The blog is presently undergoing a make-over in order to bring it up to date with the latest research.

## 2. Computing Technology / Software

Over the course of the journey, a large number of different computing systems and software have been used. In order to

[2] http://ns-waves-and-structures.blogspot.com

run *Algor*, *Star-CD* and *Flo++* a reasonably powerful desktop computer was sufficient.

Dortmund University introduced the author to *Suse* Linux, mini and mainframe computers. *Featflow* could run efficiently on reasonably-priced workstations, as well as mini, mainframe computers and even decent laptops/notebooks. At one point, an Apple iMac was converted to Linux and brought into service. Two PDA's (Personal Digital Assistants) expired in the process, however.

Chulalongkorn University used standard desktop computers, running variants of their research codes. These solver codes were optimized to perform very efficiently. This was sufficient for teaching and numerical research purposes. Compiler optimization and vectorization of the tensor-based codes often allowed for up to tenfold speed improvements. An in-house software package called *EasyFEM* provided a useful geometry builder, mesh-generator, and solver launcher. Fairly large, complex two-dimensional models could be built using this software.

During the years of self-sponsorship, various computers have been employed ranging from multi-core laptops, through to recovered and re-purposed servers e.g. HP ML350, Supermicro 1U. The secret has been to optimize the computational mesh to fit within the limitations of the available computing equipment.

As the Linux operating system has matured over the years, the model sizes have increased. In some models up to 8 Gb RAM has been used. Various flavors of Linux have been used over the years. The main constraint on model size beyond 8 Gb RAM is the CPU performance.

Recently Ubuntu 18.04 LTS server is used as the operating system of choice. It is stable and well-supported. The *lxde* graphics environment is installed, along with numerous support packages.

In later years, the project has settled on the following Open Source solver software:

- FreeFem++[3] [Hecht, 2012]
- Elmer[4]

*Elmer* is very useful in a multi-physics environment e.g. natural convection with flow and thermal coupling. The software can be downloaded, but, it appears that a Linux Mint virtual machine is preferred at this point. It eventually became difficult to build a complete installation on Ubuntu Linux, and the use of *Elmer* was discontinued.

*FreeFem++* is an extremely powerful environment for working with pde's (partial differential equations). A complex script file has been developed, which, with a few program changes, can model a number of 2D advanced geometries. Graphical output can be generated during the solution phase using Postscript files, which are then displayed by a ps/pdf reader. The major portion of data output is exported via vtk files, and visualized in either *Paraview* or *VisIt*.

[3] http://freefem.org ; http://www3.freefem.org
[4] http://csc.fi/web/elmer

The methodology used is as follows:

- Solve for $u_1$, $u_2$, $p$
- Record *time*
- Compute mathematical relationships
- Export data output e.g. direct graphics ps, or vtk
- Iterate for all time steps
- Visualize

The visualization software used in the project has settled on the following Open Source software:

- Paraview[5]
- VisIt[6] [VisIt, 2012]

The amount of visualization data generated for a single simulation run can be in the order of between 150-500 Gb, depending on the number of solver iterations, and mesh density. Thus, each run has to be compressed before storage.

The process used for capturing each simulation is as follows:

- Adjust process parameters - text editor
- Start simulation
- Screen capture to confirm mesh and process parameters
- Start *Paraview* or *VisIt*
- Load prepared graphic state
- Run through simulation output - repeat
- Simulation completes
- Generate movies - avi, mpeg
- Run movies and review at appropriate speed
- Compress data - 7z extreme compression
- Saved compressed data to long-term storage

Cloud computing was investigated but the huge amount of data transfer from the cloud back to the visualization computer, as well as costs, were considered to be challenging, but not insurmountable hurdles. Envisaged would be running *VisIt* or *Paraview* in the cloud and sending only final visualizations and movies back to the project. The costs of repatriating the data versus storage in the cloud would then be of interest.

## 3. Gallery and Discussion

### 3.1 CU-MEC3 - 2006

CU-MEC3 was a seminar arranged by Chulalongkorn University for postgraduate students to present their recent findings (refer to Figure 1). [Aubery, 2006]

The paper was titled *"A Brief Review of the Wave Nature of the Continuity and Incompressible Momentum Conservation Equations"*.

**Abstract** - The continuity and incompressible momentum equations are compared, in form, to a number of classical parabolic wave-forms, including dispersion, diffusion and forcing function terms. The fluid velocity field is decomposed into bulk flow and fluctuating flow fields subject to

[5] http://paraview.org
[6] http://wci.llnl.gov/simulation/computer-codes/visit

a momentum-closure condition. The concept of wave motion within a confined flow domain is explored. A number of numerical simulations are presented, displaying wave-like phenomena. Various postulates are presented regarding the possibility of viscous shock-wave activity within the incompressible viscous fluid flow field.

The family of first-order (in time) wave equations were explored, including: one dimensional planar wave equation; first-order linear wave equation (advection equation); inviscid Burgers (non-linear 1st-order wave equation); Burgers equation (non-linear 1st-order wave equation with diffusion); Burgers equation with source term; Korteweg de Vries (KdV) equation; incompressible momentum equations (Eulerian description) in tensor form; continuity equation; and finally the system of incompressible Navier-Stokes equations.

The relationship between the various equations was highlighted by writing each in tensor form. A review of the Penalty Method used in the *FreeFem++* solver was presented in order to explain the negligible effect that the penalty value imposed on the dynamics (refer to Figure 2).

Encouragement was drawn from White's [White, 2006] observations that the continuity, momentum and energy equations display a mix of boundary-value and wavelike characteristics, containing elliptic, parabolic and hyperbolic properties.

At this phase of the project, the possibility of low-speed viscous incompressible equivalent to the high-speed compressible shock-wave was investigated. The momentum and continuity equations were in the form of wave equations, and so more detailed wave explorations followed.

**If it looks like a wave, moves like a wave, reacts like a wave - then it may very well be a wave**. This was very much a voyage of discovery.

**Critical lines** - In Figure 3 for $u_2$, a number of thin red lines are shown, upstream and downstream of the cylinder. These represent regions in the flow field of common flow direction. Additional features emerge as the Reynolds Number (Re) rises from 175 to 263. Definite flow structures are evident.

In Figure 4 at Re=5848, a number of vertical red lines emerge in the region downstream of the cylinder. These represent zones in which the $u_2$ velocity direction alternates. The structure behind the cylinder is observed to rupture.

### 3.2 CU-MEC5 - 2007

The paper for the CU-MEC5 conference was titled *"In search of momentum waves - Part 1 - plug wave inside a pipe using a penalty-based method"* (refer to Figure 5). [Aubery, 2007]

**Abstract** - The motion of a plug wave inside a two-dimensional duct is investigated. Alternative interpretations of incompressibility are discussed, leading to a modified system of equations useful for modeling almost incompressible fluid flows. A brief introduction to wave phenomena reviews temporal and spatial variations, moving and standing wave-forms, and acceleration-force balances. A number of useful visualization techniques are presented, including scale-leveling. Numerical simulation results are presented and an equation derived for the wave group velocity as a function of distance.

The flow domain was subjected to a fixed velocity step input and the resulting plug wave progression was observed (refer to Figure 6 and Figure 7).

**High-count contour plots** - The technique of using high-count contour plots enabled various features to be observed that otherwise would have been swamped by a full-color rendering.

***log abs* plots** - Another technique that emerged during this phase was the use of the *log* of ratios of interest - the *geometric scale leveler*. This allowed scale compression to allow both small and large scale effects to be observed simultaneously. This technique has been used in various forms for the remainder of the project.

**Solver timing** - An important finding was the correct solver timing to be used in order to accurately track the front of the moving plug wave. Use of this technique has allowed stable solutions. Using this wave timing, solution divergence is an extremely rare occurrence. This timing has been used for all subsequent simulations.

### 3.3 RAEFM - 2008

The 2nd International Symposium on Recent Advances in Experimental Mechanics (RAEFM 2) took place in Vijayawada, India in 2008. [Aubery, 2008]

The conference paper was titled *"Momentum-Driven Waves - Air Flow Over Cylinder Within Confined 2D Duct"* (refer to Figure 8).

**Abstract** - The motion of air flow over a cylinder within a two-dimensional duct is investigated. Alternative interpretations of incompressibility are discussed, leading to a modified system of equations useful for modeling almost incompressible fluid flows. A brief introduction to wave phenomena reviews temporal and spatial variations, moving and standing wave-forms, acceleration-force balances, and velocity jumps. A number of useful visualization techniques are presented, including high resolution contour plots, scale-leveling and cut-plane surface visualization. Numerical simulation results are presented for the onset of unsteady flow.

The onset of instability from a Reynolds number of 57, through 60 and 63, was shown (refer to Figure 9, Figure 10 and Figure 11).

**Momentum waves** - The onset of instability was observed in a change of modal shapes from Re=57 to Re=63. These were postulated to be *momentum waves* due their occurrence in the velocity field.

**Log abs plots** - The $log|u_2/u_1|$ velocity jump plots captured the changing dynamics well, across a range of spatial scales.

During the RAEFM 2 Symposium, the author had the pleasure of meeting Prof. Kojiro Suzuki, who chaired the session where the [Aubery, 2008] paper was presented. During the session he mentioned an earlier paper in Japanese which validated the findings. [Tezuka and Suzuki, 2003]

Referring to Figure 4 (12) of the Tezuka-Suzuki paper, the similarity to Fig A1.5 $\left(\frac{\partial u_2}{\partial t}\right)$ and Fig A1.6 $\left(\frac{\partial v}{\partial t}\right)_{mag}$ was noted.

### 3.4 2008 - 2018

This period was very much a time of exploration and discovery. More than one thousand simulations were performed using *Featflow*, *FreeFem++* and *Elmer*. Various benchmarks were developed and explored - some tracing back into mid 1940's literature. In the end, *FreeFem++* has become the standard simulation environment for this project. It can run efficiently on reasonable hardware, and is generally very fast. *Elmer* was extremely useful for multi-physics applications, especially with flow and temperature coupling.

It was eventually decided to standardize on the *Featflow* benchmark, but with the cylinder placed on flow center-line. Comparative observations showed that the offset used by the *Featflow* team did nothing, in general, to disturb the flow patterns, for the diameter of cylinders interrogated. The center-line approach was considered in order to force a breaking of symmetry.

The fluid parameters used in the benchmark approach also proved to allow fast solution times - as in wall-clock times of up to 120 sec (or 180 sec in some cases). In general, in the order of 10,000-25,000 solver time points are required to adequately capture the flow dynamics. Time is required to allow the initial transients to pass through the flow domain. Thereafter, a certain settling time is required, in order to provide a suitable experimental framework.

Visualization was performed using *FreeFem++*, *Elmer*, *Paraview*, and *VisIt*.

The initial data capture approach was to have *FreeFem++* print out postscript files at each computation time-step. This proved to be very slow, however. Orders of magnitude improvement were obtained by writing the data into vtk files at each time-step, then visualizing these in parallel using *Paraview* and *VisIt*.

A sample of plots generated during the 2008-2018 period are shown. Refer to Figures 13, 14, 15, 16. 17, 18, 19, 20, 21, 22, 23, 24, 25, 26, 27, 28, 29. (Clicking on a reference will jump to the appropriate figure).

**Traveling transverse waves** - A number of the plots provide evidence of *traveling transverse waves*, in the downstream portion of the flow field.

**Velocity, acceleration at Re=150** - Refer to Figures 13, 14,15. The acceleration plots show that the vortices shed from the rear of the cylinder begin to form themselves into a triangular structure.

**Angle between velocity & acceleration vectors** - Refer to Figure 16. Relationships appear between the velocity and acceleration vectors. Note the circular patterns towards the rear portion of the flow field.

**Velocity waves** - Refer to Figures 17, 18, 19, 20. Wave structures can clearly be seen in the 2nd order velocity wave plot. The 3rd order velocity wave structure appears similar to that of vorticity.

**Pressure waves** - Refer to Figure 21. The pressure wave is a transverse wave. This transverse nature can also be observed in Figure 9 (Fig A1.7) for $\frac{\partial p}{\partial t}$.

**Vorticity, log abs vorticity & vorticity waves** - Refer to Figures 22,23,24,25. The 1st order vorticity wave - Figure 23 has wavelike structure.

**Streamlines** - Refer to Figure 26 The streamlines have a definite structure. Mass confinement between the cells is clear. The cells are convected down the downstream field. If a vertical observation line is taken at a point in the downstream flow field, the field will appear to oscillate vertically, as the streamline cells pass through the observation point. This is further evidence of the transverse nature of the traveling waves.

**Viscous dissipation** - Refer to Figures 27, 28. Viscous dissipation plots reveal where energy is dissipated in the flow field. The *log abs* scale-leveling technique shows details across small-to-large scales. The hexagonal structures correspond to similar structures in the acceleration (force) field.

**Plane waves in vorticity field** - Refer to Figure 29. This is a fascinating plot in that it connects plane waves and vorticity structures. The 'bubble' immediately in front of the three cylinders oscillates vertically over time - striking out waves as it does so. The plane waves appear to pass either through the centers of the vortices, or reasonably close to them.

### 3.5 2019 - present

This period has brought a great deal of consolidation to the project. Multiple servers/workstations were employed in order to accelerate the research on a 24/7 basis.

A sample of plots generated during the period 2019-present are shown. Refer to Figures 30, 31, 32, 33, 34, 35, 36, 37, 38, 39, 40, 41, 42, 43, 44, 45, 46, 47, 48, 49, 50, 51, 52. (Clicking on a reference will jump to the appropriate figure).

Around this time, *Paraview* began to add in some new visualization features which allowed visual determination of the point at which the onset of instability emerges in a 2D flow field - the critical Reynolds number (Re,c). At present, a long series of simulations is in progress to determine Re,c for various diameters of the circular obstruction, using the classical *Featflow/Elmer* benchmark geometry as the basis. An accuracy for Re,c to one decimal place has been achieved thus far.

An exciting point was reached in the when the author discovered the paper - *"Modal Analysis of Fluid Flows: An Overview"* [AIAAJ, 2017] which reviewed various mathematical techniques enabling flow modes to be determined in the wake of flows over wings and aerofoils, providing both frequency and amplitude. A series of videos of the conference session were also made available on *Youtube*. The session was extremely informative.

During one of the talks, mention was made, in passing, of traveling wave solutions in fluid flow systems. From all appearances, these were not high-speed, compressible flows. This was a critical moment for this project. **Traveling waves were (almost) acknowledged in (incompressible) fluid flows**.

What followed very shortly thereafter were a series of discoveries by the author, of additional wave activity upstream of the flow obstruction. These show promise in explaining the physical mechanisms that cause flow instability to occur.

**Traveling transverse waves** - In the above-mentioned series of pictures, a number of traveling wave types are observed. Transverse velocity waves (1st order) in Figure 39. Transverse velocity waves (2nd order) in Figure 43. Transverse pressure waves (1st order) in Figure 36. Transverse pressure waves (2nd order) in Figure 42.

**Wave and singularity surface indicators** - A number of wave and singular-surface indicators have been used in this project: *vjumpL*, *ajumpL*, *a2jumpL*, *aljumpL*, *al2jumpL*. Each generates lines where each of the parameters of interest tend to zero - the 'critical lines'. Of interest are also conditions where both terms tend to zero simultaneously. This occurs where critical lines of different types cross, leading to a singularity in the flow field. Vortex sheets and shear layers can be seen in *vjumpL* in red.

## 4. Papers in Progress

In parallel with ongoing simulation and data collection, a number of new papers are in preparation. These draw on the academic and visualization research of the period 2008-2019, as well as recent findings.

**Determination of critical Reynolds Number for flow over cylinder in 2D duct** - The critical Reynolds' Number (Re,c) for a range of cylinder diameters (D) is determined. A plot of the Re,c-D is shown. Novel wave-based techniques are used to arrive at a high level of accuracy.

**Vorticity analysis** - High-resolution patterns behind a cylinder in 2D duct flow. Critical lines are developed.

**Traveling transverse waves** - Various wave-forms are presented and analyzed. $u_1$, $u_2$, $Vmag$, $\frac{\partial v}{\partial t}$, $\frac{\partial p}{\partial t}$.

**Higher-order waves** - Higher-order forms of transverse velocity waves, transverse pressure waves, transverse acceleration waves, and transverse jerk waves.

**Wave and singular surface indicators** - Review of *vjumpL*, *ajumpL*, *a2jumpL*, *aljumpL*, *al2jumpL*, *jljumpL*, and their associated 'critical lines'.

**Acceleration-Force fields** - Acceleration-force fields are visualized.

**Viscous dissipation** - Viscous dissipation fields are visualized.

**Nabla v tensor** - Theoretical analysis and visualization of the nabla velocity ($\nabla v$) tensor.

**Where does the boundary layer go?** - Visualization of the boundary layer from a tensor-analysis perspective.

**Dynamic Mode Decomposition** - The use of modern DMD and derivative numerical techniques to determine modal amplitudes and frequencies.

**Singular surfaces** - Truesdell et al discuss singular surfaces of various orders [Truesdell and Toupin, 1960] [Truesdell and Rajagopal, 2000] [Truesdell and Noll, 2004]. A selection of these, including acceleration waves, are investigated.

**How to...** - Practical techniques to explore the fundamental theories developed during the project. This may become a series of papers.

**State of...** - A compendium summarizing the work done to date, by this project, and other authors. Summary of our collective ability to explain the observed physical phenomena. How close are we to understanding the fundamental phenomena that cause turbulence?

## 5. Going Forwards

The discoveries made over the life of the project have been substantial. The incompressible Navier-Stokes equations have begun giving up some of their long-held secrets. The experimental observations are in accordance with continuum mechanics theory for singular surfaces [Truesdell and Toupin, 1960] [Truesdell and Rajagopal, 2000] [Truesdell and Noll, 2004].

Project highlights include:

- Traveling transverse waves are evident in the wakes of flow obstructions
  - Transverse velocity waves
  - Transverse pressure waves
  - Transverse acceleration waves
- 'Critical lines' exist in the velocity field
  - Where $u_1$ tends to zero
  - Where $u_2$ tends to zero
  - Where both $u_1$ and $u_2$ tend to zero simultaneously
  - Identify regions of common velocity direction
  - Identify shear layers
- A number of 'wave indicators' have been developed
  - Velocity fields, acceleration fields, jerk fields
  - Identify first and higher-order wave-forms
- Wave activity also occurs upstream of the obstacle
- The Re,c for 2D flow over a cylinder in a duct (*Featflow/Elmer* benchmark), has been determined as a function of diameter
- Singular surfaces of high order are observed in incompressible fluid flows
- A wave-based theory has been developed to provide understanding for:
  - Reynolds number
  - Strouhal number
- Theory to determine the angle between various vectors e.g. between velocity and acceleration using flow-field scalar values

### 5.1 Expanding the project

The findings thus far have the project on a trajectory to explain possible mechanisms underlying the turbulence phenomenon.

In order to accelerate progress on this project, it would be of great benefit to open up to additional academic collaborators. More computational resources are definitely required. The open topics can feed a hungry academic team for quite some time.

In addition, the techniques used can be expanded to other experimental areas, for instance PIV (Particle Image Velocimetry) and newer technologies . If $u_1$, $u_2$ and *time* are captured, the computational and visualization techniques can easily be applied.

## Statement

We are stewards of the discoveries that we may unearth during the journeys we make. We do not own these discoveries, nor may we selfishly hide them from others, for self-gratification. We are responsible to bring these into the world for others to enjoy.

## Acknowledgments

The author would like to express his sincere thanks to the many colleagues who have debated the possible existence of waves in incompressible fluid flows. Some debates were fierce, others less so, and a select few (one perhaps) who believed this to be at all possible.

A special 'thank you' to, *Tom* (internet discussion), Prof. Stefan Turek (*Featflow/Feast* and team), the professors at Chulalongkorn (for their endless patience), Prof. Suzuki, Prof. Rathakrishnan, Prof. Frederic Hecht (*FreeFem++*), *Elmer* developers, *Paraview* and *VisIt* developers, as well as the *Ubuntu* and *lxde* developers.

The adventure continues...

## References

[White, 2006] White, M. W. (2006). Viscous Fluid Flow, 3 ed. McGraw-Hill, International Edition.

[Aubery, 2006] Aubery, D. (2006). A brief review of the wave nature of the continuity and incompressible momentum conservation equations. *3rd CU-MEC.*

[Aubery, 2007] Aubery, D. (2007). In search of momentum waves, Part 1 - plug wave inside a pipe using a penalty-based method. *5th CU-MEC.*

[Aubery, 2008] Aubery, D. (2008). Momentum-driven waves - Airflow over cylinder confined within 2D duct. *The Second International Symposium on Recent Advances in Experimental Mechanics (RAEFM 2).*

[Tezuka and Suzuki, 2003] Tezuka, A. and Suzuki, K. (2003). Global Stability Analysis of Asymmetric Flow around a Two-Dimensional Cylinder, a Sphere, and a Three-Dimensional Low Aspect Ratio Cylinder. *JSSAS, Vol. 51, No. 597, pp. 529-536, 2003 (Japanese).*

[Truesdell and Toupin, 1960] Truesdell C. and Toupin R.A. (1960). The Classical Field Theories. Encyclopedia of Physics, Vol. III/1, pages 226-858. Springer-Verlag.

[Truesdell and Rajagopal, 2000] Truesdell, C and Rajagopal K.R. (2000). An Introduction to the Mechanics of Fluids. Birkhäuser Boston.

[Truesdell and Noll, 2004] Truesdell C. and Noll W. (2004,1992,1965). The Non-Linear Field Theories of Mechanics, 3 ed. Springer-Verlag.

[AIAAJ, 2017] Taira K., Brunton S.L., Dawson S.T.M., Rowley C.W., Colonius T., McKeon B.J., Schmidt O.T., Gordeyev S., Theofilis V. and Ukeiley L.S. (2003). Modal Analysis of Fluid Flows: An Overview. *AIAA Journal, Vol. 55, No. 12, December 2017.* https://arc.aiaa.ord/doi/10.2514/1.J056060 ; https://doi.org/10.2514/1.J056060

[Hecht, 2012] Hecht F. (2012). New development in FreeFem++. *Journal of Numerical Mathematics, Vol. 20, No. 3-4, pp. 251–265, 2012.*

[VisIt, 2012] Childs H., Brugger E., Whitlock B., Meredith J., Ahern S., Pugmire D., Biagas K., Miller M., Harrison C., Weber G.H., Krishnan H., Fogal T., Sanderson A., Garth C., Bethel E.W., Camp D., Rübel O., Durant M., Favre J.M., Navrátil P. VisIt: An End-User Tool For Visualizing and Analyzing Very Large Data. *High Performance Visualization–Enabling Extreme-Scale Scientific Insight, Oct, pp. 357-372, 2012.*

The references cited are a tiny fraction of the books reviewed during the project, but they are particularly relevant to the present project historical review. Copies of the author's papers can be requested per e-mail.

# A Brief Review of the Wave Nature of the Continuity & Incompressible Momentum Conservation Equations

Desmond Aubery

Dept. of Mechanical Engineering
Chulalongkorn University
Bangkok 10330, Thailand
E-mail: *Desmond.A@student.chula.ac.th*

The continuity and incompressible momentum equations are compared, in form, to a number of classical parabolic waveforms, including dispersion, diffusion & forcing function terms. The fluid velocity field is decomposed into bulk flow and fluctuating flow fields subject to a momentum-closure condition. The concept of wave motion within a confined flow domain is explored. A number of numerical simulations are presented, displaying wave-like phenomena. Various postulates are presented regarding the possibility of viscous shock-wave activity within the incompressible viscous fluid flow field.

**Key Words: Navier-Stokes, parabolic traveling wave, momentum waves**

## 1. Introduction

It is common experience that numeric solvers for CFD (computational fluid dynamics) simulations are subject to solution divergence (blow-up) if certain critical parameters are exceeded. The underlying physical reasons for this blow-up are not clear. Use is made of numeric stabilization mechanisms to enhance solution stability with various forms of convection stabilization schemes being used in most commercial CFD algorithms. These schemes have been shown to affect solution accuracy [4].

In order to attempt to more fully understand the underlying mechanisms at work within confined fluid domains, this paper reviews the possibility of wave solutions for the incompressible forms of the continuity & momentum conservation equations for viscous fluid flow. Initial research findings are provided in order to develop an understanding of wave nature of these equations. This paper is intended to be exploratory in nature.

A brief review is provided of a number of typical first-order wave partial differential equations (pde's) [3,5,9,11,12]. The Navier-Stokes equations for incompressible fluid flow are then decomposed, and the resultant velocity field equations compared in form to these known wave equations. The final section provides a number of numeric flow simulations, display wave-like phenomena.

The mathematical existence of traveling wave solutions for nonlinear parabolic systems of pde's is reviewed extensively for combustion & chemical reactions, by Volpert [9], with the introduction of min/max estimates of wave velocity and use of wave bifurcation theory to provide information on the bifurcated waveforms occurring after the collapse of the initial planar wave form. The study of nonlinear wave phenomena is an emerging research area.

An alternative title for this paper could be: "Can incompressible fluids exhibit wave phenomena when flowing in confined domains?"

## 2. Forms of typical first-order wave pde's

A number of classic waveforms are reviewed in order to present the basic form & structure of typical first order wave equations [1, 3, 8, 9]. 1D, and equivalent derivative, tensor symbolic & tensor indicial forms (here restricted to 1D, for form comparison purposes) are presented for completeness.

**One dimensional planar wave equation**

$$\frac{\partial u}{\partial t} - \alpha . \frac{\partial^2 u}{\partial x^2} = f(u) \qquad (2.1.1)$$

$$u_t - \alpha \, u_{xx} = f(u) \qquad (2.1.2)$$

$$\frac{\partial \mathbf{v}}{\partial t} - \alpha \, \nabla^2 \mathbf{v} = f(\mathbf{v}) \qquad (2.1.3)$$

$$\frac{\partial v_i}{\partial t} - \alpha \, v_{i,jj} = f(v_i) \qquad i, j = 1 \quad (2.1.4)$$

This equation is 1st-order in time, 2nd-order in space and describes a traveling wave, where $f(u)$ represents a forcing-function, or source term, and $\alpha$ is a constant.

Draft - 08.09.2006, Rev A - 22.03.2007

**Figure 1.** CU-MEC3 - Front page

## 21. Appendices

### 21.1. Penalty form of Navier-Stokes equations

In this section, an *almost* incompressible form of the Navier-Stokes is derived, in a similar manner to standard incompressible formulations. We begin with the compressible momentum equation in tensor form & then insert the derivative of the velocity divergence.

The compressible form of the momentum equation, in tensor indicial form, is given by [1]:

$$\frac{\partial v_i}{\partial t} + v_j v_{i,j} - \nu\, v_{i,jj} - \frac{1}{3}\nu\, v_{j,ji} = -\frac{1}{\rho} p_{,i} + a_i \quad \textit{(21.1.1)}$$

The penalty method sets the divergence of the velocity as being proportional to the thermodynamic pressure, in symbolic tensor form as follows,

$$\nabla \bullet \mathbf{v} = \varepsilon\, p \quad \textit{(21.1.2)}$$

Or, in indicial tensor form, as,

$$v_{i,i} = \varepsilon\, p \qquad i = 1,2,3 \quad \textit{(21.1.3)}$$

Differentiate eqn *(21.1.3)* with respect to spatial coordinates,

$$(v_{i,i})_{,j} = (\varepsilon\, p)_{,j}$$
$$v_{i,ij} = \varepsilon\, p_{,j}$$

Swap indices *i* & *j*,

$$v_{j,ji} = \varepsilon\, p_{,i} \qquad i = 1,2,3 \quad \textit{(21.1.4)}$$

$$\frac{1}{3}\nu\, v_{j,ji} = \frac{1}{3}\nu\,\varepsilon\, p_{,i} \quad \textit{(21.1.5)}$$

Insert eqn *(21.1.5)* into eqn *(21.1.1)*,

$$\frac{\partial v_i}{\partial t} + v_j v_{i,j} - \nu\, v_{i,jj} - \frac{1}{3}\nu\,\varepsilon\, p_{,i} = -\frac{1}{\rho} p_{,i} + a_i$$

$$\frac{\partial v_i}{\partial t} + v_j v_{i,j} - \nu\, v_{i,jj} = -\left(\frac{1}{\rho} - \frac{1}{3}\nu\,\varepsilon\right) p_{,i} + a_i$$

Inserting, $\nu = \frac{\mu}{\rho}$

$$\frac{\partial v_i}{\partial t} + v_j v_{i,j} - \nu\, v_{i,jj} = -\left(1 - \frac{1}{3}\mu\,\varepsilon\right)\frac{1}{\rho} p_{,i} + a_i \quad \textit{(21.1.6)}$$

Define,

$$\frac{1}{\hat{\rho}} = \left(1 - \frac{1}{3}\mu\,\varepsilon\right)\frac{1}{\rho} \quad \textit{(21.1.7)}$$

Thus, eqn *(21.1.6)* becomes,

$$\frac{\partial v_i}{\partial t} + v_j v_{i,j} - \nu\, v_{i,jj} = -\frac{1}{\hat{\rho}} p_{,i} + a_i \quad \textit{(21.1.8)}$$

Using a typical fluid viscosity ( $\mu$ ) for water of 855e-3 kg/ms, and penalty parameter ( $\varepsilon$ ) of 1e-6, the density multiplier in eqn *(21.1.7)* becomes,

$$\left(1 - \frac{1}{3}\mu\,\varepsilon\right) = 1 - \frac{1}{3}.(855e-3).(1e-6) \approx 1$$

In practice, for water and air,

$$\frac{1}{\hat{\rho}} = \left(1 - \frac{1}{3}\mu\,\varepsilon\right)\frac{1}{\rho} \approx \frac{1}{\rho}$$

The Navier-Stokes system of equations may now be written, in indicial tensor form, as,

$$\boxed{\frac{\partial v_i}{\partial t} + v_j v_{i,j} - \nu\, v_{i,jj} = -\frac{1}{\hat{\rho}} p_{,i} + a_i} \quad \textit{(21.1.8)}$$

$$\boxed{v_{i,i} = \varepsilon\, p} \qquad i = 1,2,3 \quad \textit{(21.1.3)}$$

Or, in symbolic tensor form as,

$$\boxed{\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \bullet \nabla \mathbf{v} - \nu\, \nabla^2 \mathbf{v} = -\frac{1}{\hat{\rho}} \nabla p + \mathbf{a}} \quad \textit{(21.1.9)}$$

$$\boxed{\nabla \bullet \mathbf{v} = \varepsilon\, p} \quad \textit{(21.1.2)}$$

Thus, for water and air, the penalty method produces the familiar incompressible form of the momentum equation, with the continuity equation now becoming proportional to thermodynamic pressure, instead of being uniquely zero.

Draft - 22.03.2007

**Figure 2.** CU-MEC3 - Appendix 21.1 - Review of Penalty Method

**15.2.3. Flow over cylinder within a pipe**

| | | |
|---|---|---|
| Pipe dimensions | : | 285 x 25 mm (modeled as 0.285 x 0.0250 m).<br>cylinder diameter. 5 mm; positioned 0.1125 m from inlet. |
| Fluid | : | water. |
| Kinematic viscosity | : | $\nu = 855e-9\_m^2/s$ . |
| Inlet velocity | : | various. |
| Comment | : | EasyFEM CBS solver [13]; approx. 80,000 P1 triangular elements; unstructured; prescribed outlet condition. |

- **Inlet velocity** **:** $u_{1,in} = 0.006\_m/s$ **(*Re*$_{in}$ = 175.44, *Re*$_{cyl}$ = 35.09)**

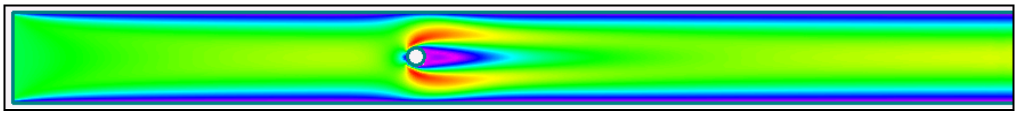

**Figure 15.2.3.1 – Velocity magnitude**

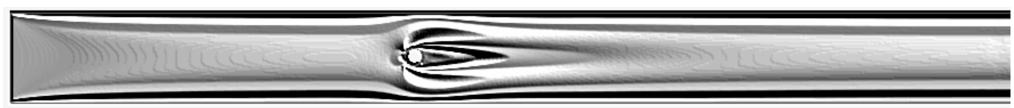

**Figure 15.2.3.2(a) – Velocity magnitude – embossed to enhance flow structures**

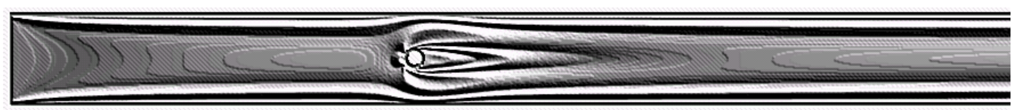

**Figure 15.2.3.2(b) – Velocity magnitude – embossed to enhance flow structures (16-bit)**

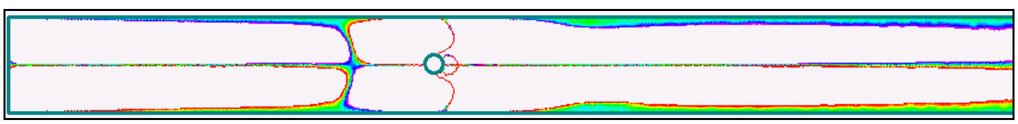

**Figure 15.2.3.3 –** $u_2$ **(*v*) velocity – data range ±1e-6 m/s**

- **Inlet velocity** **:** $u_{1,in} = 0.009\_m/s$ **(*Re*$_{in}$ = 263.16, *Re*$_{cyl}$ = 52.63)**

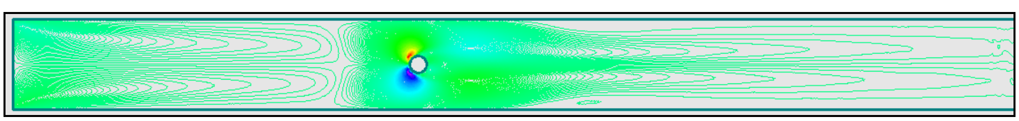

**Figure 15.2.3.4 –** $u_2$ **(*v*) velocity – data range ±1e-2 m/s**

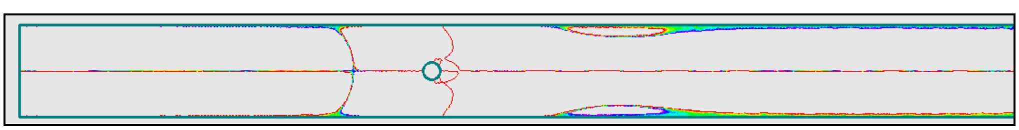

**Figure 15.2.3.5 –** $u_2$ **(*v*) velocity – data range ±1e-6 m/s**

Draft – 08.09.2006, Rev A – 22.03.2007

**Figure 3.** CU-MEC3 - Appendix 15.2.3 - Flow over cylinder within a pipe (a)

- **Inlet velocity** : $u_{1,in} = 0.2\_m/s$ **($Re_{in}$ = 5847.95, $Re_{cyl}$ = 1169.59)**

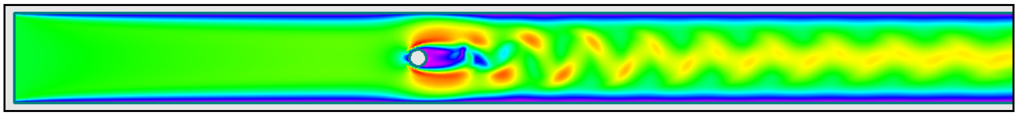

**Figure 15.2.3.6 – Velocity magnitude**

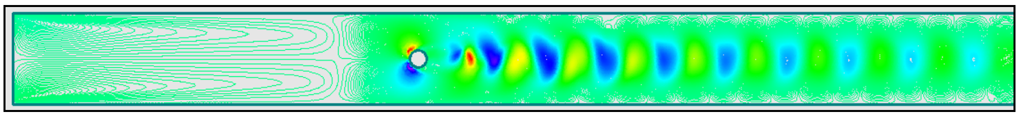

**Figure 15.2.3.6 –** $u_2$ **(*v*) velocity – data range ±1e-2 m/s**

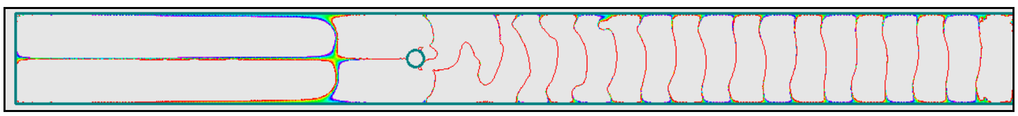

**Figure 15.2.3.7 –** $u_2$ **(*v*) velocity – data range ±1e-6 m/s**

---

Draft – 08.09.2006, Rev A – 22.03.2007

**Figure 4.** CU-MEC3 - Appendix 15.2.3 - Flow over a cylinder within a pipe (b)

# In search of momentum waves
# Part 1 - plug wave inside a pipe using a penalty-based method

Desmond Aubery

Dept. of Mechanical Engineering
Chulalongkorn University
Bangkok 10330, Thailand
E-mail: *Desmond.A@student.chula.ac.th*

The motion of a plug wave inside a two-dimensional duct is investigated. Alternative interpretations of incompressibility are discussed, leading to a modified system of equations useful for modeling almost incompressible fluid flows. A brief introduction to wave phenomena reviews temporal & spatial variations, moving & standing wave-forms, & acceleration-force balances. A number of useful visualization techniques are presented, including scale-leveling. Numerical simulation results are presented and an equation derived for the wave group velocity as a function of distance.

**Key Words: Navier-Stokes, momentum waves, penalty method**

## 1. Introduction

In the previous paper presented at 3rd CU-MEC [12], the wave nature of the Navier-Stokes equations were reviewed in form & supported by a number of numeric simulations presenting wave-like phenomena for incompressible fluids flowing within confined domains. The existence of wave phenomena at extremely low flow velocities was proposed, defining these as momentum-waves [6].

The present paper explores the momentum-wave hypothesis in greater detail and presents the numerical simulation of a plug wave inside a two-dimensional duct.

Alternative interpretations of the definition of the term *incompressibility* are discussed, resulting in the derivation of a modified system of equations used for modeling *almost* incompressible flow phenomena.

A brief introduction to wave phenomena reviews temporal & spatial variations, moving & standing wave-forms, as well as the observation that, in order to understand system dynamics, the flow must be viewed in terms of acceleration-force balances. A number of useful visualization techniques are presented.

The appendix provides graphical results from the numerical simulation of a plug wave traveling down a pipe.

## 2. System of incompressible Navier-Stokes equations

The Navier-Stokes equations for incompressible (constant property) fluids (Eulerian description), in symbolic tensor form are expressed as [1, 3, 4, 12]:

$$\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \bullet \nabla \mathbf{v} - \nu\, \nabla^2 \mathbf{v} = -\frac{1}{\rho} \nabla p + \mathbf{a} \qquad \textit{(1.1a)}$$

$$\nabla \bullet \mathbf{v} = 0 \qquad \textit{(1.1b)}$$

Or, in indicial notation as:

$$\frac{\partial v_i}{\partial t} + v_j v_{i,j} - \nu\, v_{i,jj} = -\frac{1}{\rho} p_{,i} + a_i \qquad \textit{(1.2a)}$$

$$v_{i,i} = 0 \qquad i, j = 1,2,3 \qquad \textit{(1.2b)}$$

The compact form of the momentum equation in symbolic notation is [12]:

$$\frac{\partial \mathbf{v}}{\partial t} + \left(\mathbf{v} - \nu\, \nabla\right) \bullet \nabla \mathbf{v} = -\frac{1}{\rho} \nabla p + \mathbf{a} \qquad \textit{(1.3a)}$$

Or, in indicial form,

$$\frac{\partial v_i}{\partial t} + \left(v_j - \nu\, v_{,j}\right) v_{i,j} = -\frac{1}{\rho} p_{,i} + a_i \qquad i, j = 1,2,3 \qquad \textit{(1.3b)}$$

Draft - 22.03.2007

**Figure 5.** CU-MEC5 - Front page

**21.3. Numeric Simulation Results**

**21.3.1. Results - plug wave flow in a pipe – single step**

**21.3.1.1. Sample simulation plots**

Iteration : 400
Simulation time : 0.865925 s

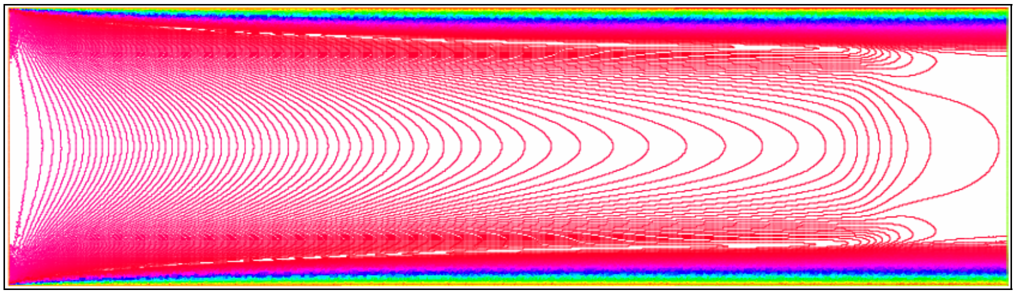

**Figure 21.3.1.1.1 –** $v_m$ **velocity [m/s]**

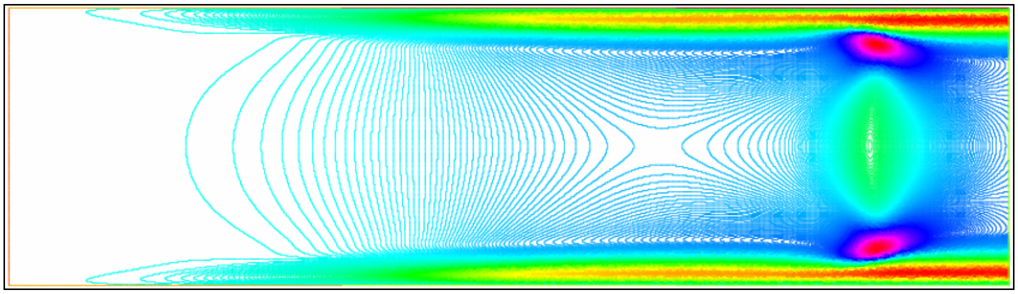

**Figure 21.3.1.1.2 –** $\partial u_1/\partial t$ **acceleration [m/s$^2$]**

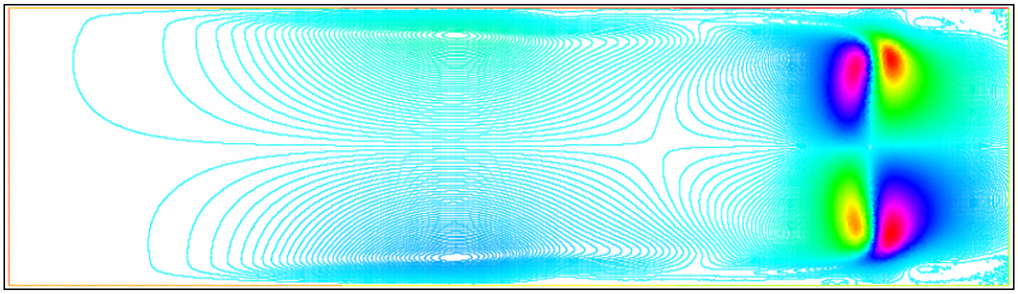

**Figure 21.3.1.1.3 -** $\partial u_2/\partial t$ **acceleration [m/s$^2$]**

Draft – 22.03.2007

**Figure 6.** CU-MEC5 - Appendix 21.3 - Numeric Simulation Results - part 1

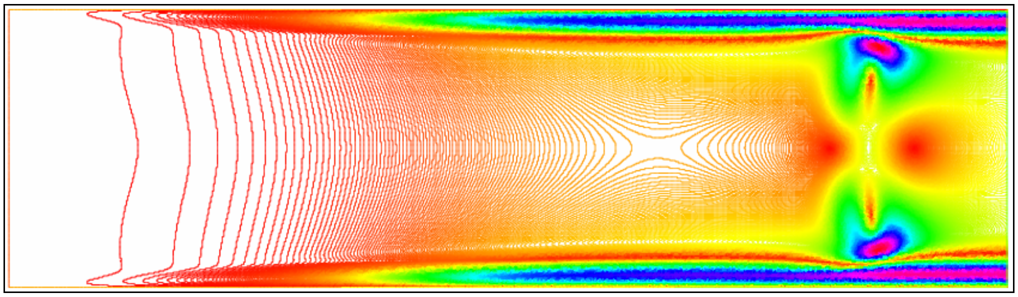

**Figure 21.3.1.1.4 –** $(\partial v / \partial t)_{mag}$ **acceleration [m/s²]**

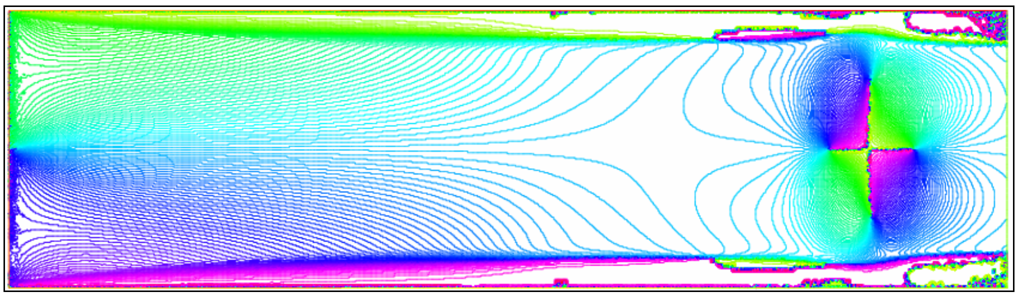

**Figure 21.3.1.1.4 –** $(\partial v / \partial t)_{ang}$ **phase angle [rad]**

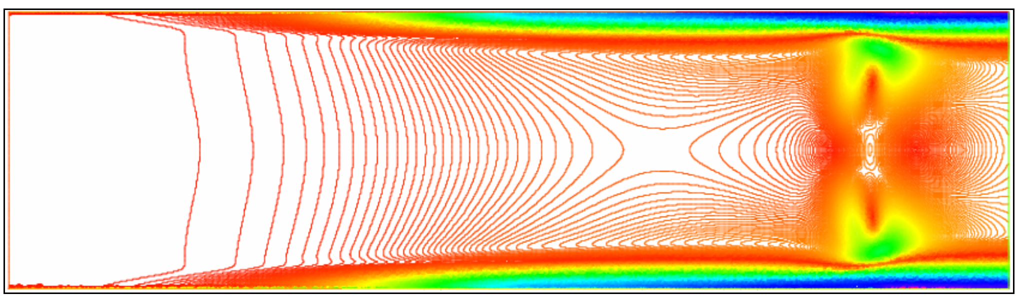

**Figure 21.3.1.1.5 –** $r_{avt}$ **ratio [1/s]**

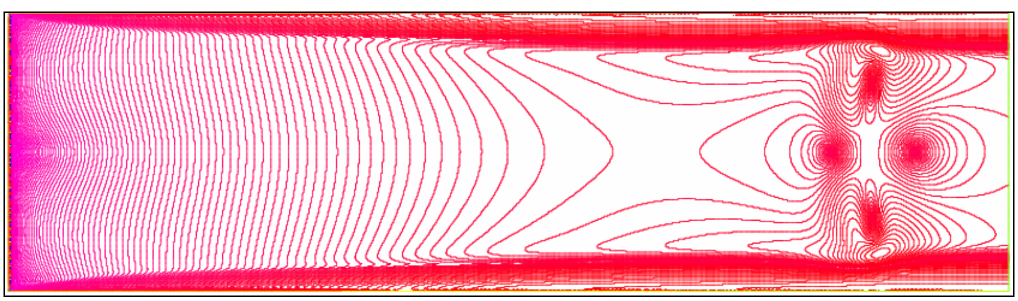

**Figure 21.3.1.1.6 –** $\log |r_{avt}|$

Draft – 22.03.2007

**Figure 7.** CU-MEC5 - Appendix 21.3 - Numeric Simulation Results - part 2

# MOMENTUM-DRIVEN WAVES
# AIR FLOW OVER CYLINDER WITHIN CONFINED 2D DUCT

D. Aubery[1]

**ABSTRACT**

The motion of air flow over a cylinder within a two-dimensional duct is investigated. Alternative interpretations of incompressibility are discussed, leading to a modified system of equations useful for modeling almost incompressible fluid flows. A brief introduction to wave phenomena reviews temporal & spatial variations, moving & standing wave-forms, acceleration-force balances, & velocity jumps. A number of useful visualization techniques are presented, including high resolution contour plots, scale-leveling & cut-plane surface visualization. Numerical simulation results are presented for the onset of unsteady flow

**Keywords: Navier-Stokes, momentum-driven waves, penalty method**

## 1. INTRODUCTION

The wave nature of the Navier-Stokes equations, presenting wave-like phenomena for incompressible fluids flowing within confined domains, were reviewed in [12,13] & supported by a number of numeric simulations. The existence of wave-like phenomena at extremely low flow velocities was proposed, defining these as momentum-waves [6].

The present paper explores the momentum-wave hypothesis in greater detail and presents the numerical simulation of air flowing over a cylinder within a confined two-dimensional duct.

Alternative interpretations of the definition of the term incompressibility are reviewed, resulting in the derivation of a modified system of equations used for modeling almost incompressible flow phenomena.

A brief introduction to wave phenomena, reviews temporal & spatial variations, moving & standing wave-forms, as well as the observation that, in order to understand system dynamics, the flow should be viewed in terms of acceleration-force balances. A number of useful visualization techniques are presented.

The appendix provides graphical results from the numerical simulation of air flow over a cylinder within a confined two-dimensional duct, over the velocity range 1.80 to 2.00 m/s, representing the onset of unsteady flow, for the particular geometry considered.

## 2. SYSTEM OF INCOMPRESSIBLE NAVIER STOKES EQUATIONS

The Navier-Stokes equations for incompressible (constant property) fluids (Eulerian description), in symbolic tensor form are expressed as [1,3,4,12]:

$$\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \bullet \nabla \mathbf{v} - \nu \nabla^2 \mathbf{v} = -\frac{1}{\rho} \nabla p + \mathbf{a} \qquad \textit{(2.1a)}$$

$$\nabla \bullet \mathbf{v} = 0 \qquad \textit{(2.1b)}$$

Or, in indicial notation as:

$$\frac{\partial v_i}{\partial t} + v_j \, v_{i,j} - \nu \, v_{i,jj} = -\frac{1}{\rho} p_{,i} + a_i \qquad \textit{(2.2a)}$$

[1] Corresponding author: Research Director, adTherm Asia, Co. Ltd., Bangkok, Thailand. email: auberydes@gmail.com,

**Figure 8.** RAEFM - Front page

**Appendix A – Numerical simulation results**

**A1. Velocity 1.80 m/s – $Re_d = 56.64$**

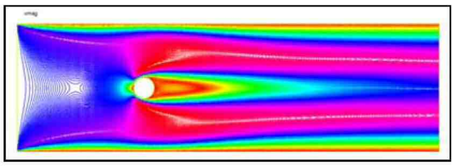

**Fig A1.1 - Velocity magnitude**

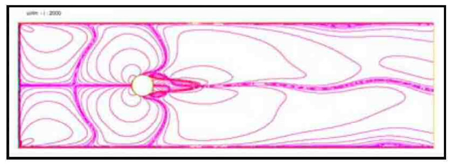

**Fig A1.2 -** $\log|u_2/u_1|$ **- velocity jumps**

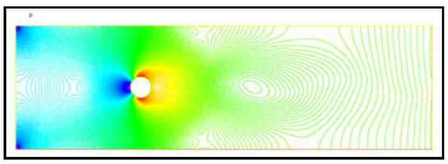

**Fig A1.3 - Pressure**

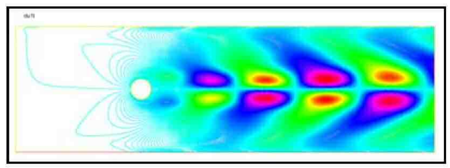

**Fig A1.4 -** $(\partial u_1/\partial t)$

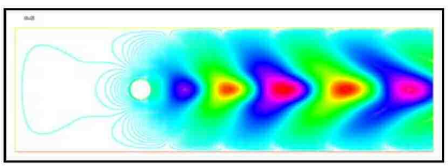

**Fig A1.5 -** $(\partial u_2/\partial t)$

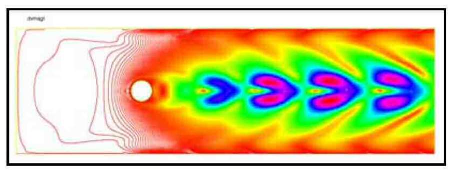

**Fig A1.6 -** $(\partial \mathbf{v}/\partial t)_{mag}$

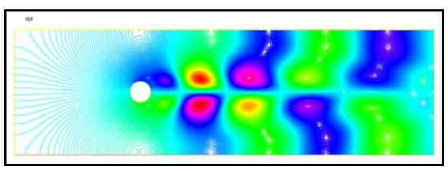

**Fig A1.7 -** $(\partial p/\partial t)$

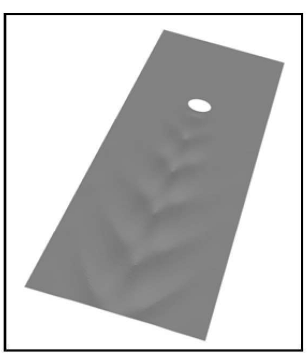

**Fig A1.8 -** $(\partial \mathbf{v}/\partial t)_{mag}$ **surface plot**

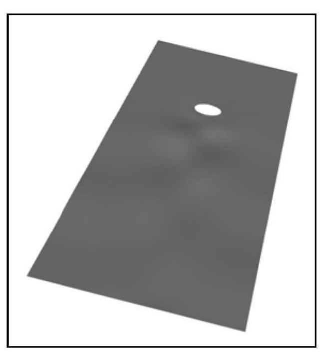

**Fig A1.9 -** $(\partial p/\partial t)$ **surface plot**



**Figure 9.** RAEFM - Appendix A-1

**A2. <u>Velocity 1.90 m/s</u> – $Re_d$ = 59.79**

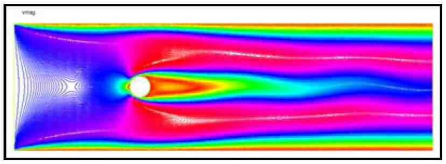

**Fig A2.1 - Velocity magnitude**

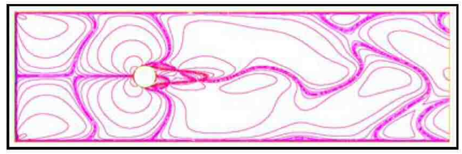

**Fig A2.2 -** $\log|u_2/u_1|$ **- velocity jumps**

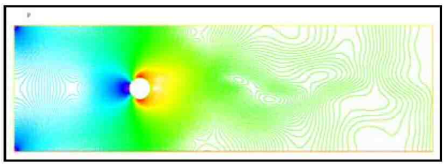

**Fig A2.3 - Pressure**

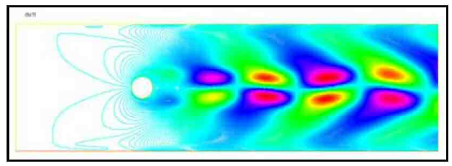

**Fig A2.4 -** $(\partial u_1/\partial t)$

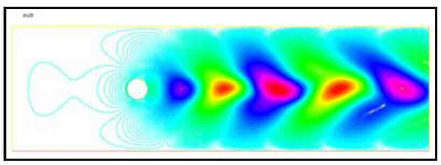

**Fig A2.5 -** $(\partial u_2/\partial t)$

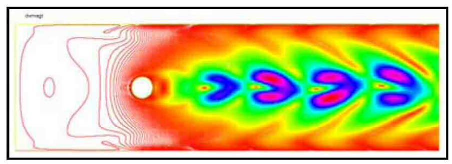

**Fig A2.6 -** $(\partial \mathbf{v}/\partial t)_{mag}$

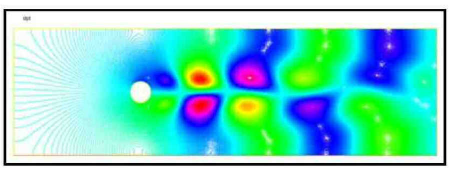

**Fig A2.7 -** $(\partial p/\partial t)$

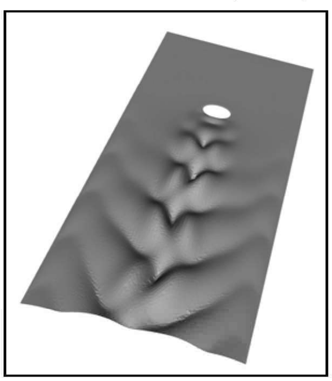

**Fig A2.8 -** $(\partial \mathbf{v}/\partial t)_{mag}$ **surface plot**

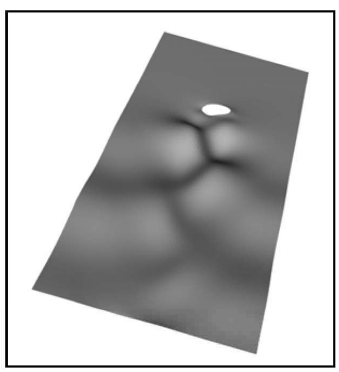

**Fig A2.9 -** $(\partial p/\partial t)$ **surface plot**



**Figure 10.** RAEFM - Appendix A-2

**A3. <u>Velocity 2.00 m/s</u> – Re$_d$ = 62.93**

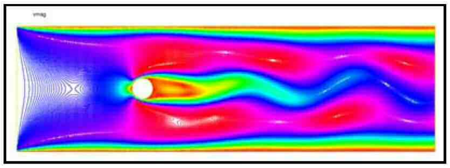

**Fig A3.1 - Velocity magnitude**

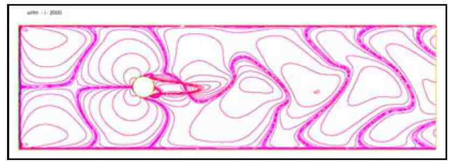

**Fig A3.2 -** $\log|u_2/u_1|$ **- velocity jumps**

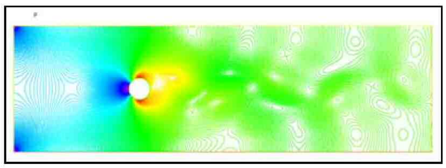

**Fig A3.3 - Pressure**

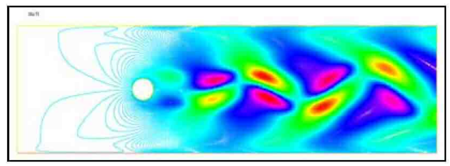

**Fig A3.4 -** $(\partial u_1/\partial t)$

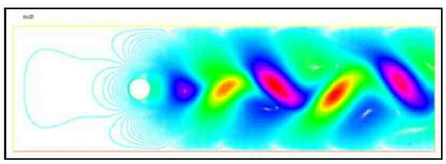

**Fig A3.5 -** $(\partial u_2/\partial t)$

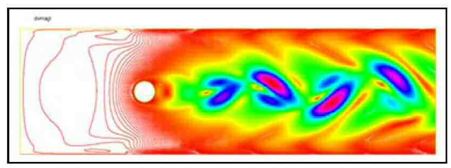

**Fig A3.6 -** $(\partial \mathbf{v}/\partial t)_{mag}$

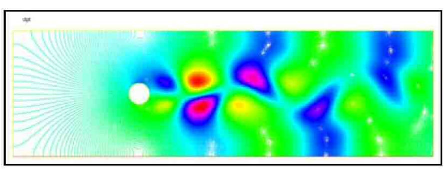

**Fig A3.7 -** $(\partial p/\partial t)$

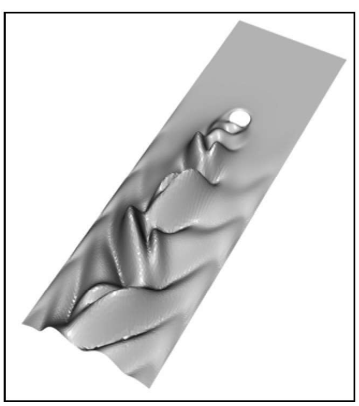

**Fig A3.8 -** $(\partial \mathbf{v}/\partial t)_{mag}$ **surface plot**

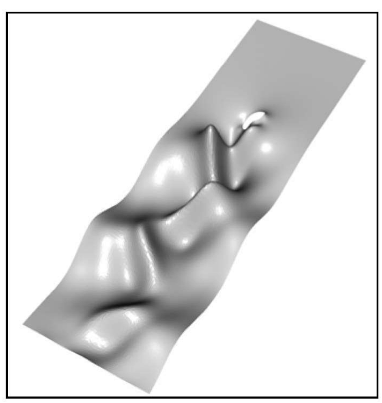

**Fig A3.9 -** $(\partial p/\partial t)$ **surface plot**



**Figure 11.** RAEFM - Appendix A-3

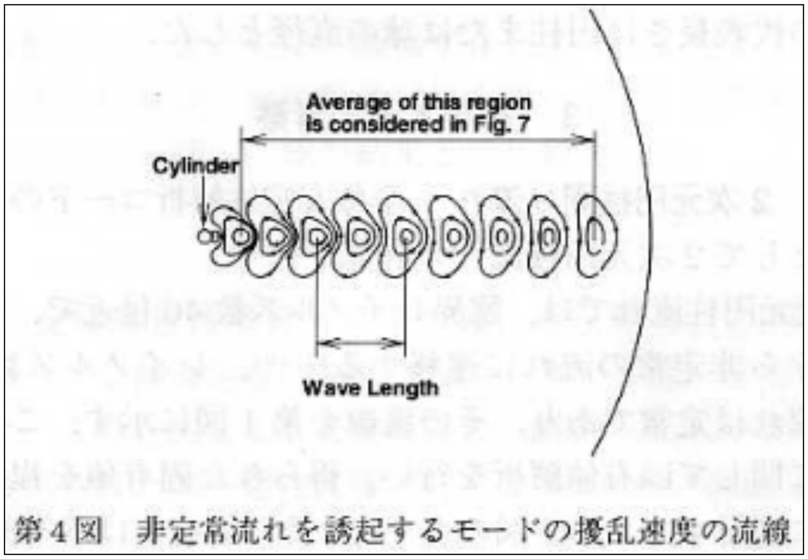


**Figure 12.** Tezuka, Suzuki - Fig 4

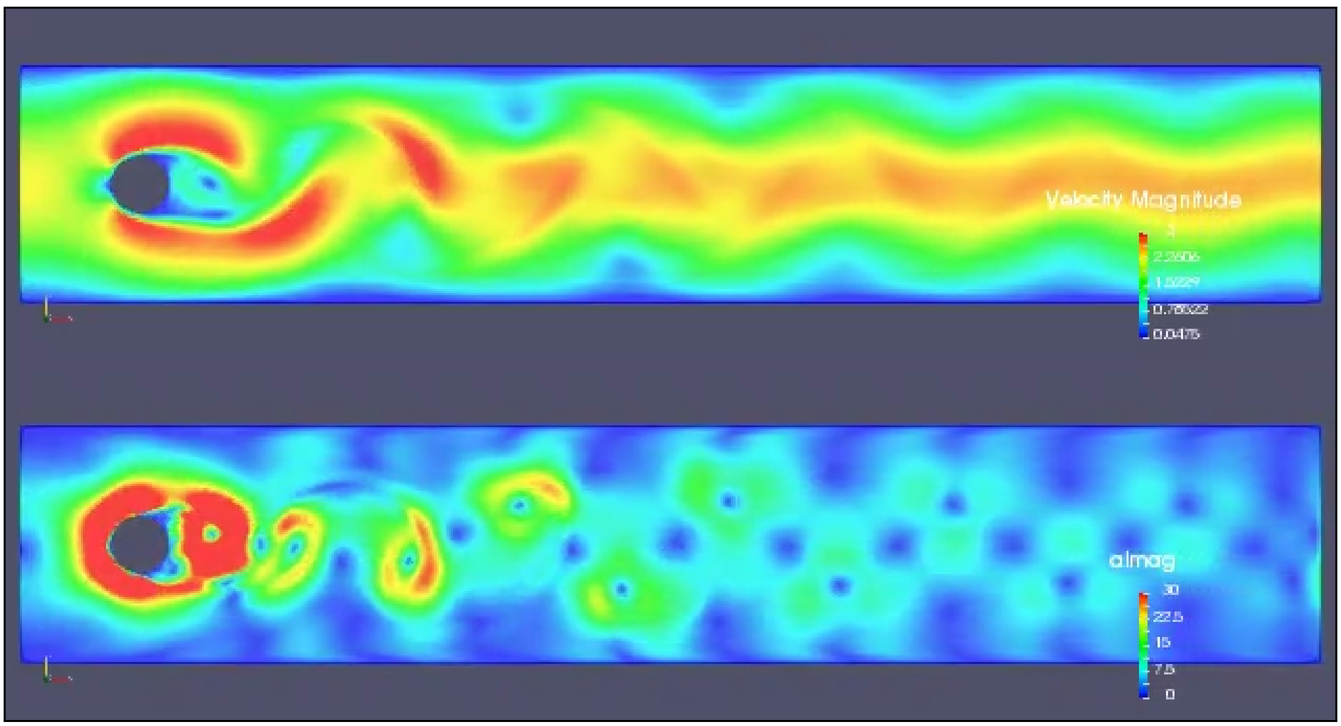


**Figure 13.** Velocity, acceleration at Re=150

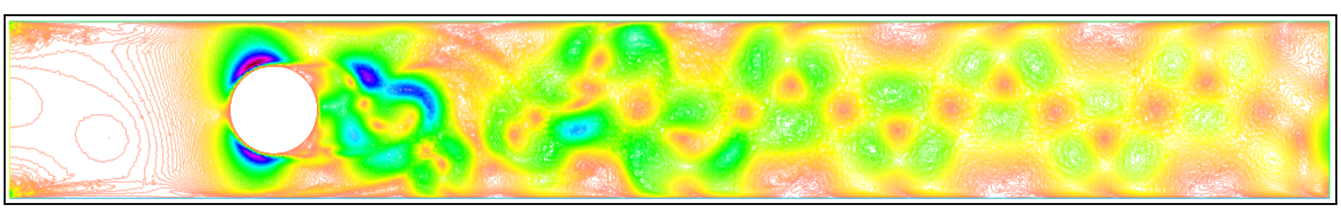

**Figure 14.** Acceleration - structures

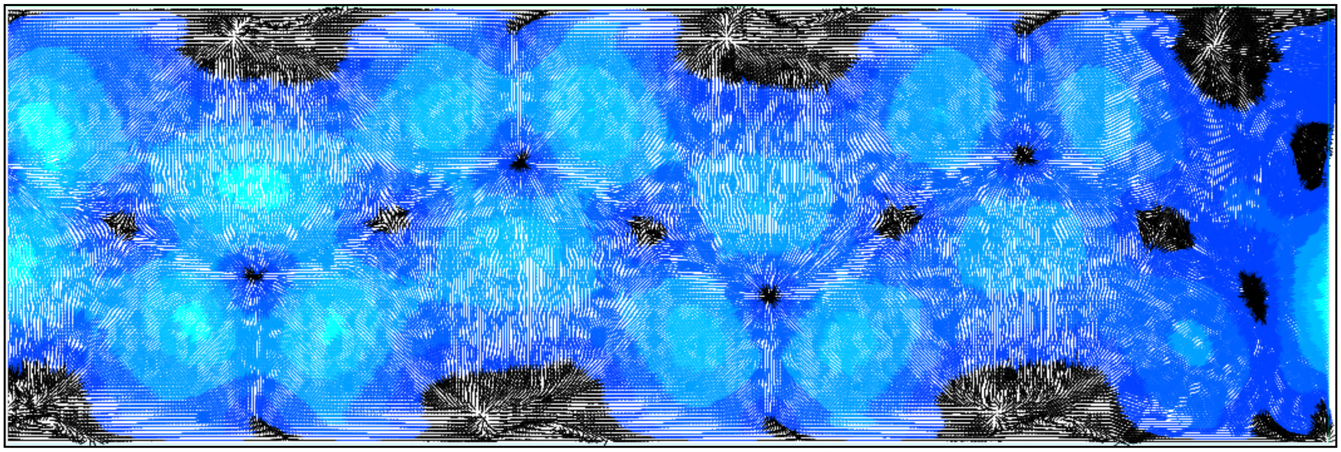

**Figure 15.** Acceleration - vector field

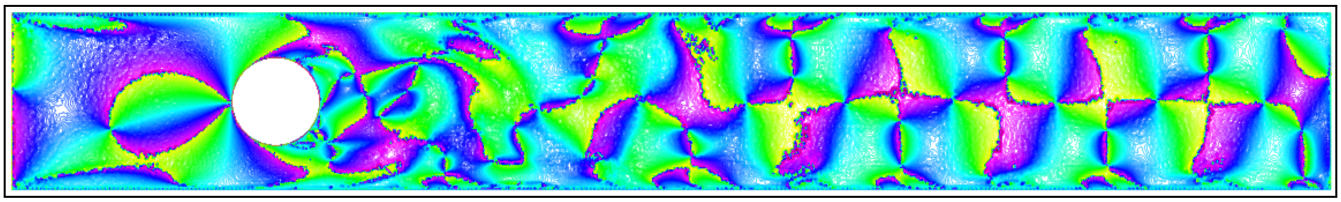

**Figure 16.** Angle between velocity and acceleration vectors

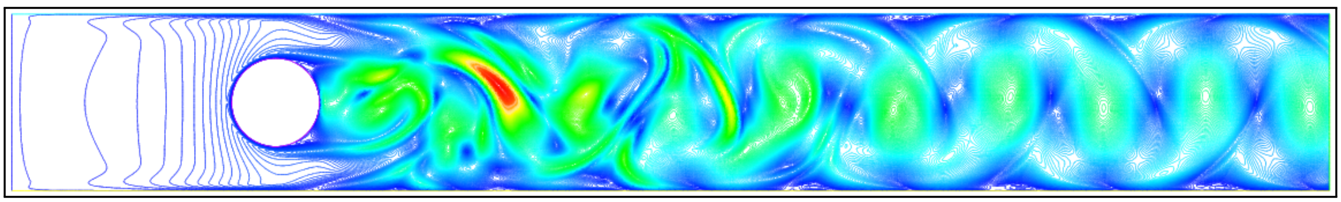

**Figure 17.** Velocity wave - 1st order

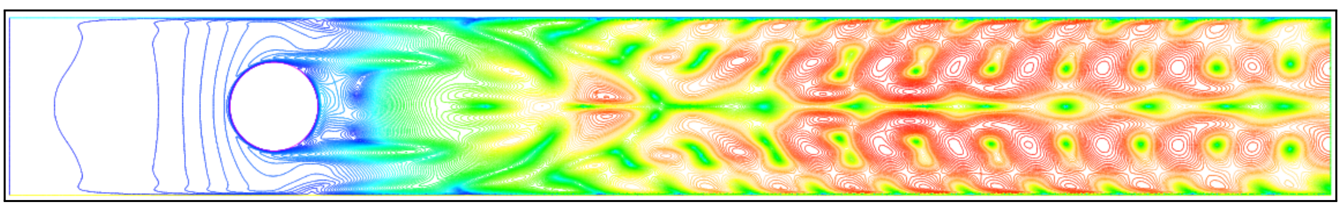

**Figure 18.** Velocity wave - 2nd order

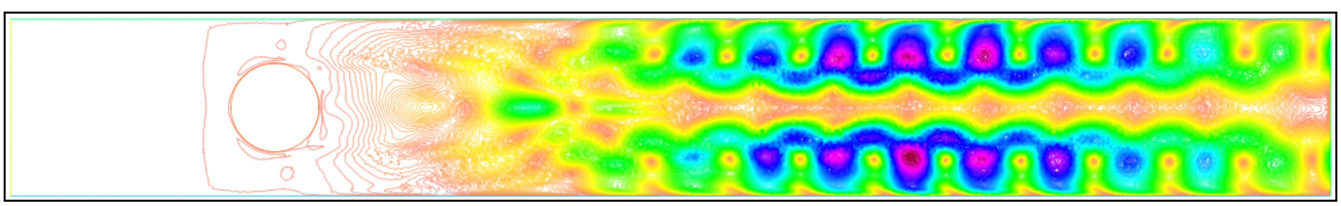

**Figure 19.** Velocity wave - 2nd order

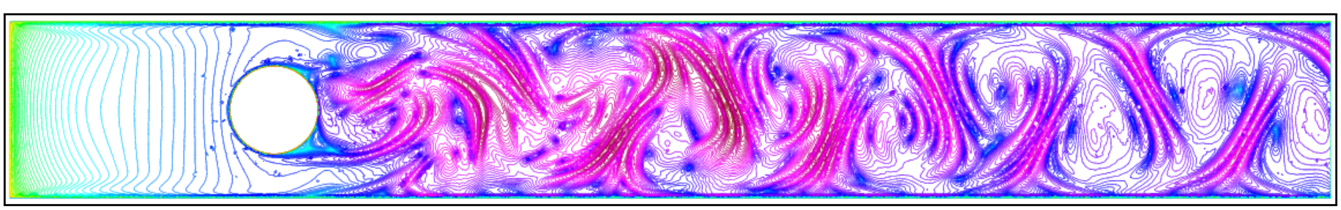

**Figure 20.** Velocity wave - 3rd order

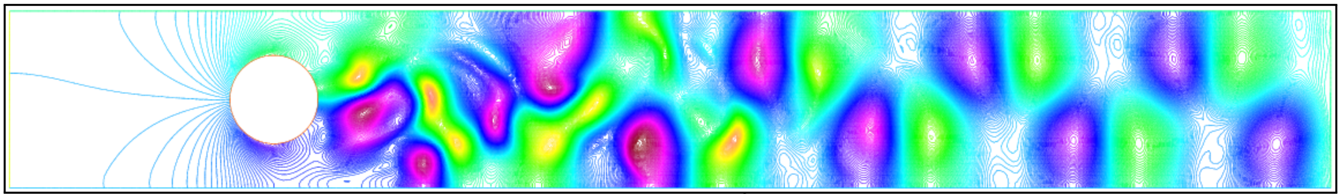

**Figure 21.** Pressure wave ($\frac{\partial p}{\partial t}$) - 1st order

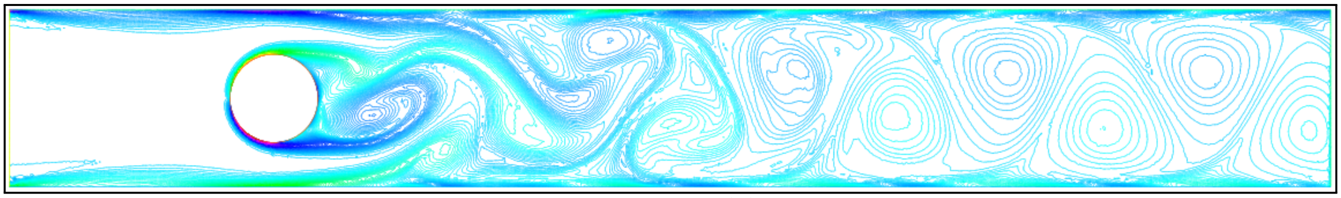

**Figure 22.** Vorticity

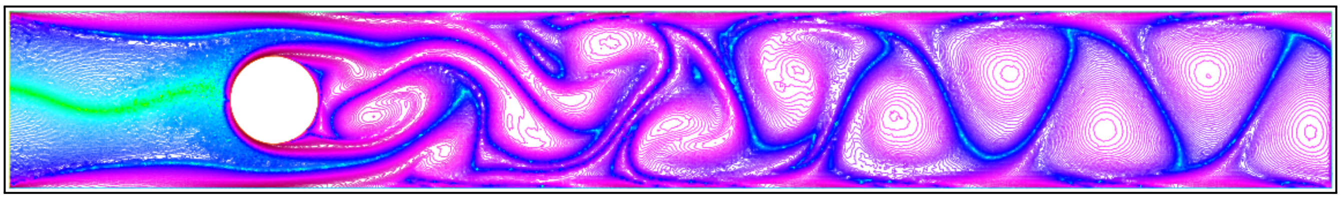

**Figure 23.** log abs vorticity

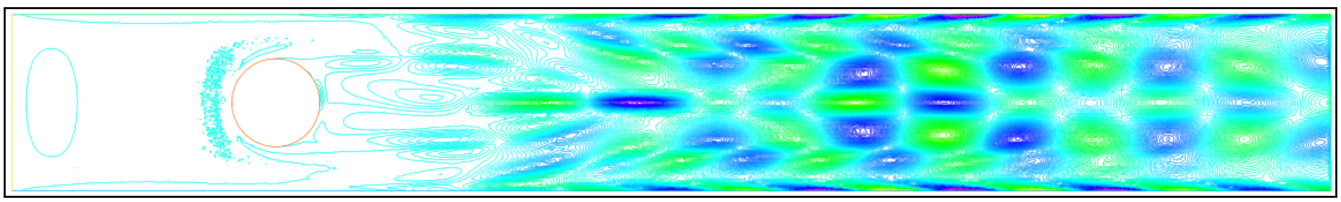

**Figure 24.** Vorticity wave - 1st order

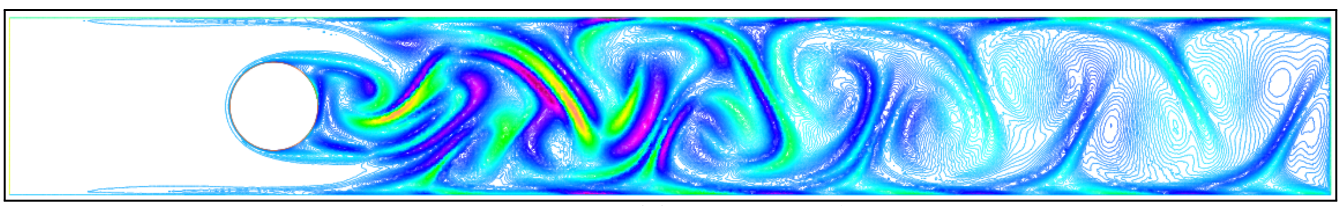

**Figure 25.** Vorticity wave - 1st order

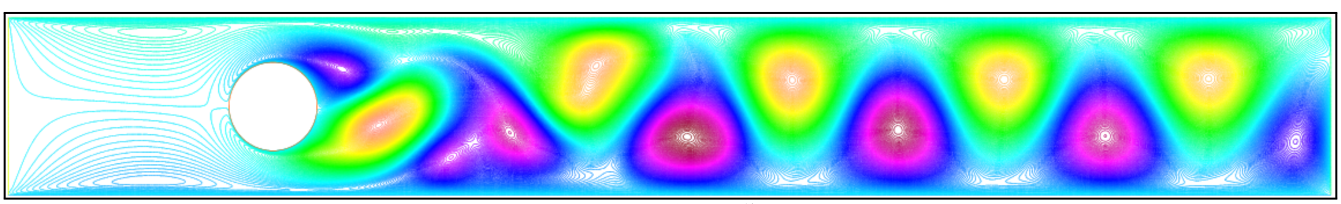

**Figure 26.** Streamlines

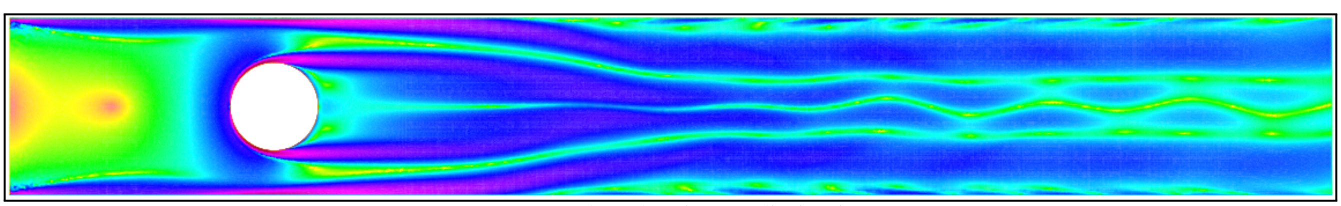

**Figure 27.** log abs viscous dissipation

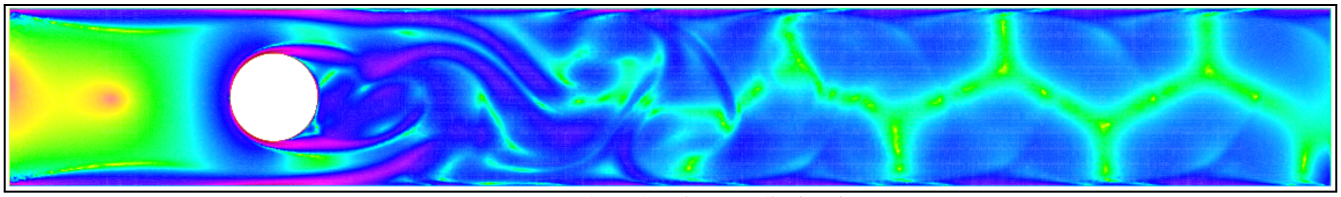

**Figure 28.** log abs viscous dissipation

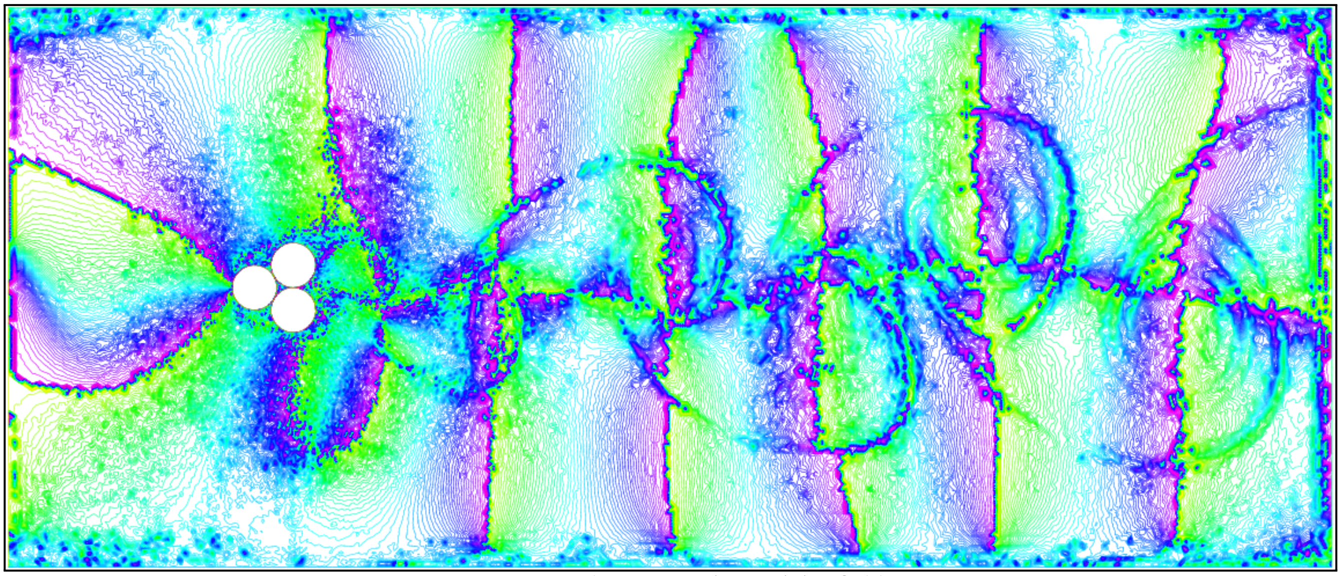

**Figure 29.** Plane waves in vorticity field

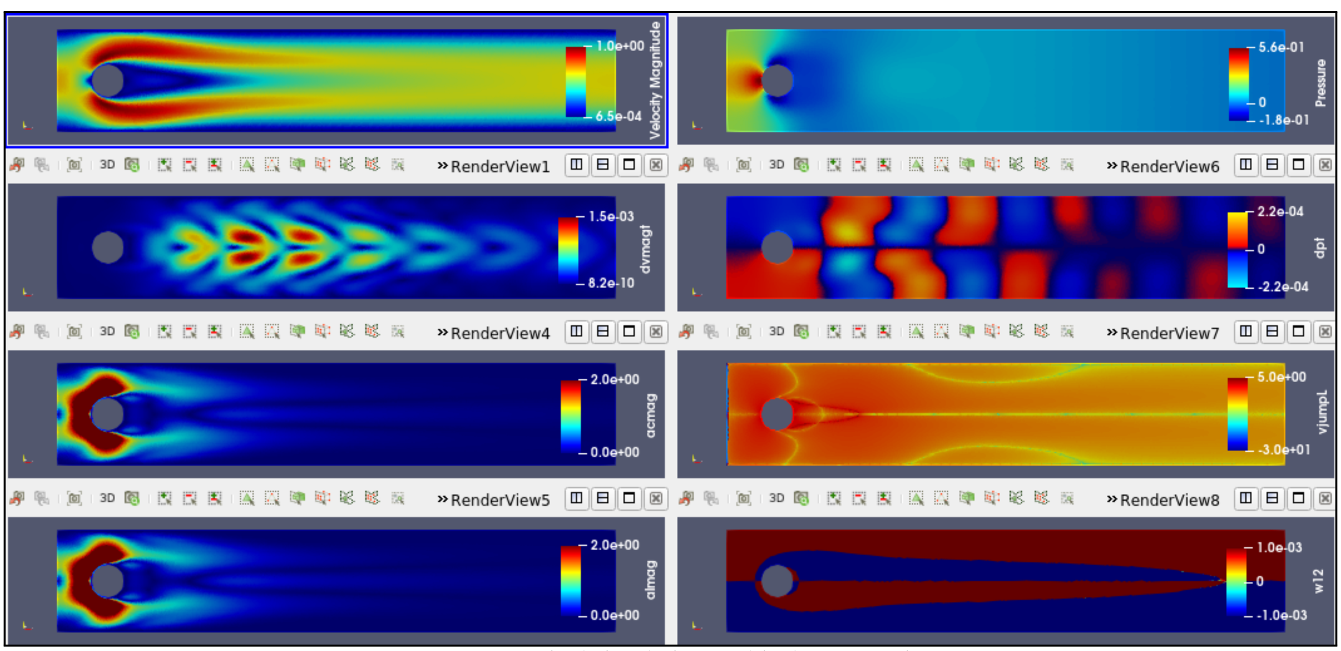


**Figure 30.** Typical simulation multi-plots (Paraview)

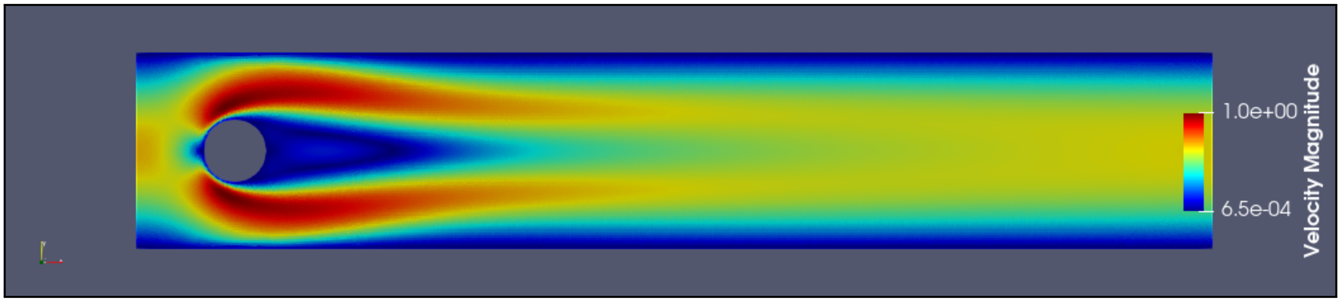


**Figure 31.** Velocity magnitude

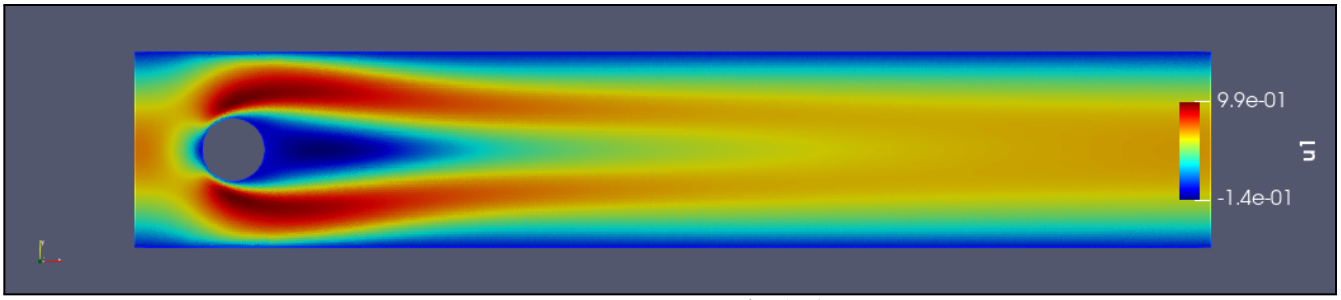


**Figure 32.** x-component of velocity $u_1$

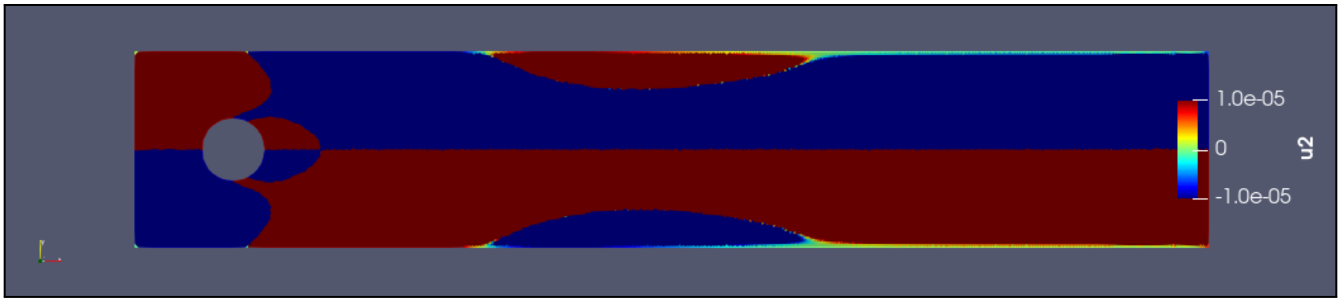


**Figure 33.** y-component of velocity $u_2$

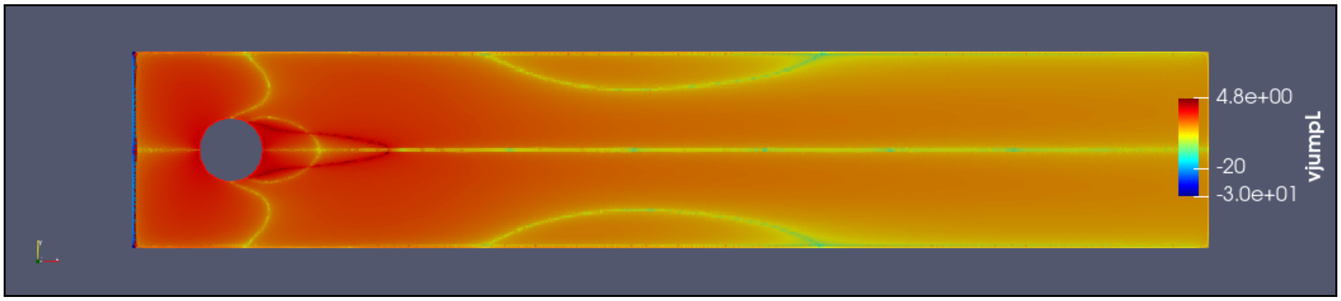


**Figure 34.** Velocity jump - *vjumpL*


**Figure 35.** Pressure

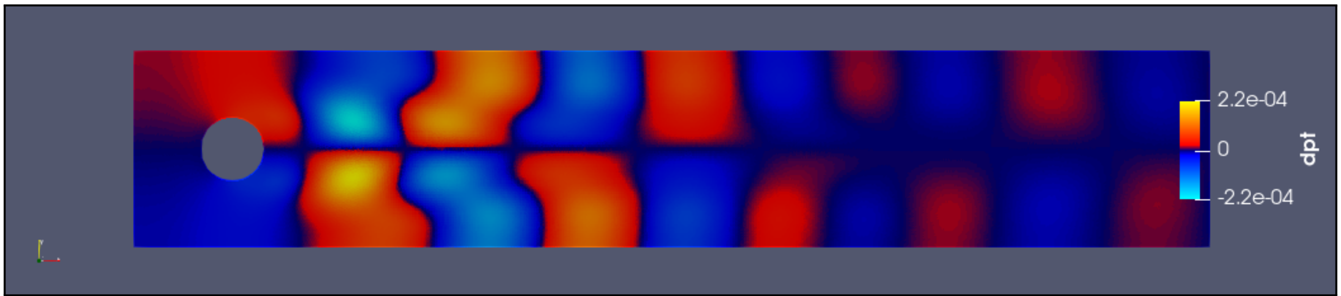


**Figure 36.** Transverse Pressure wave - 1st order - $\frac{\partial p}{\partial t}$

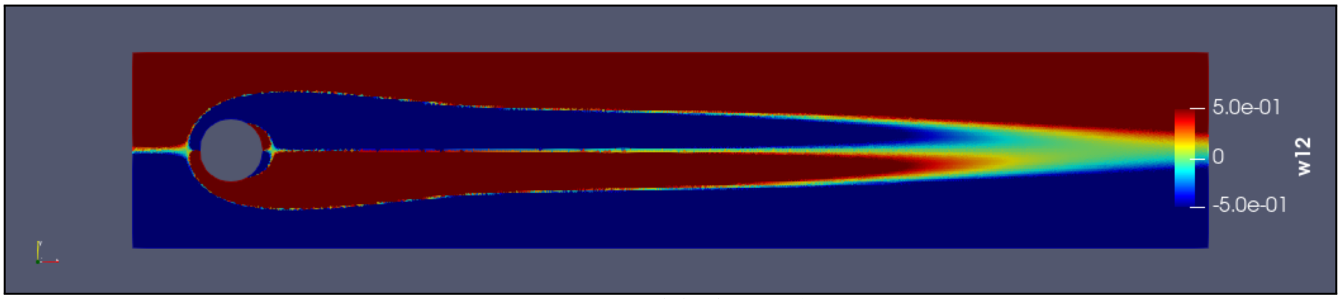


**Figure 37.** Vorticity in 1-2 plane

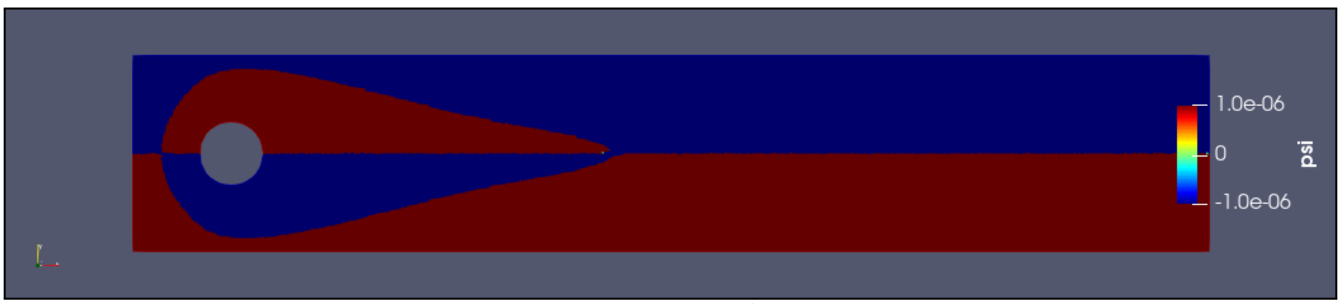


**Figure 38.** Streamlines

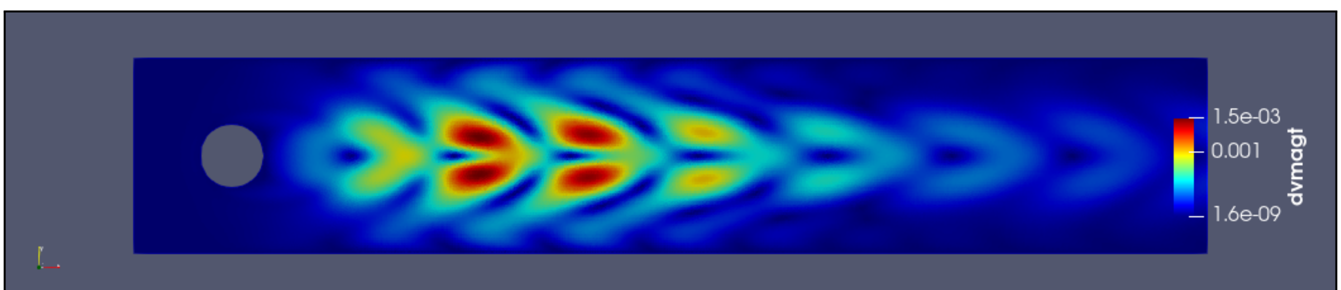


**Figure 39.** Velocity wave - 1st order - $\frac{\partial v}{\partial t}$


**Figure 40.** Convection acceleration

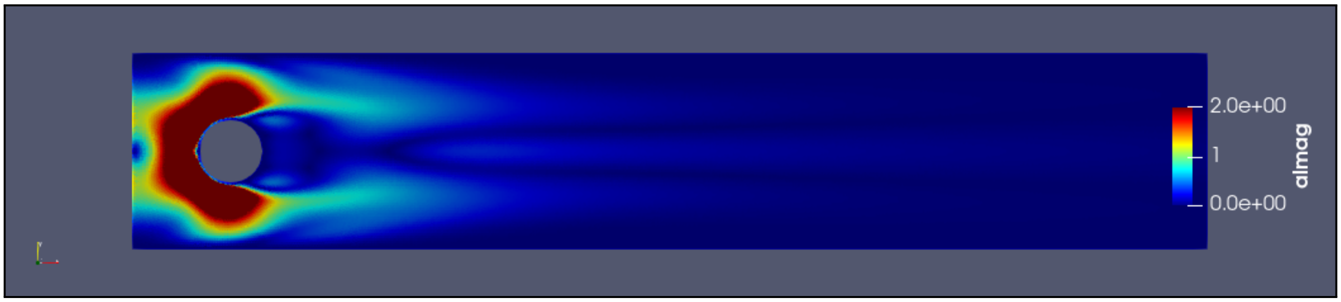


**Figure 41.** Acceleration

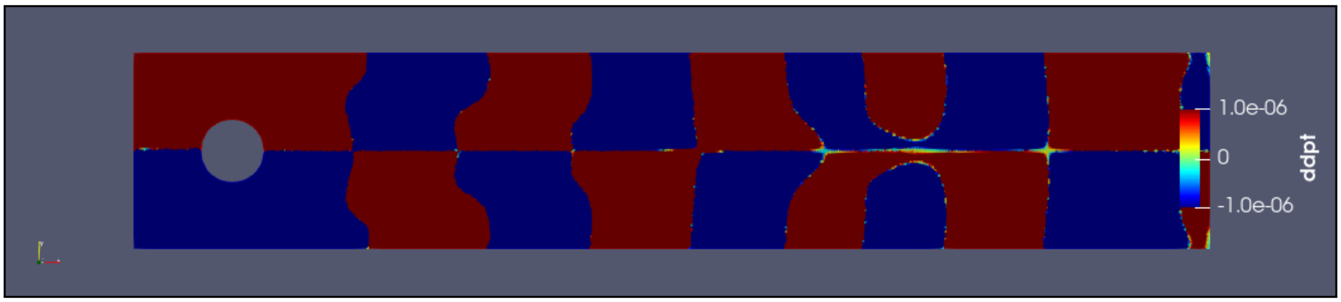


**Figure 42.** Transverse pressure wave - 2nd order

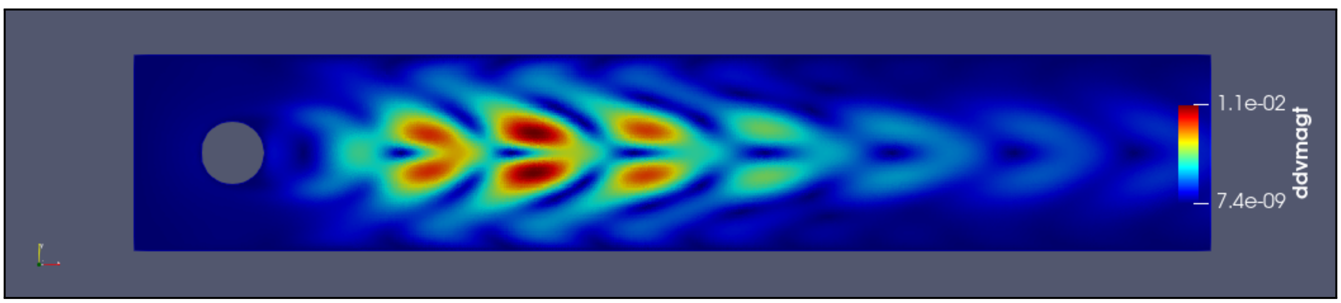


**Figure 43.** Velocity wave - 2nd order

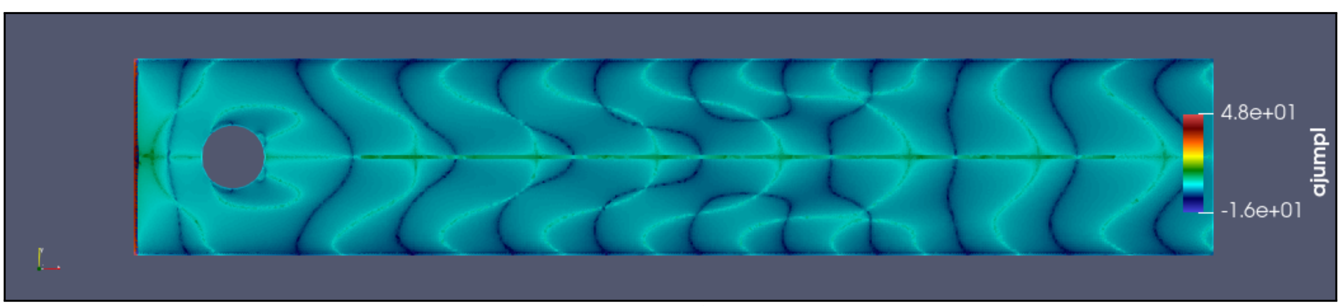


**Figure 44.** ajumpL

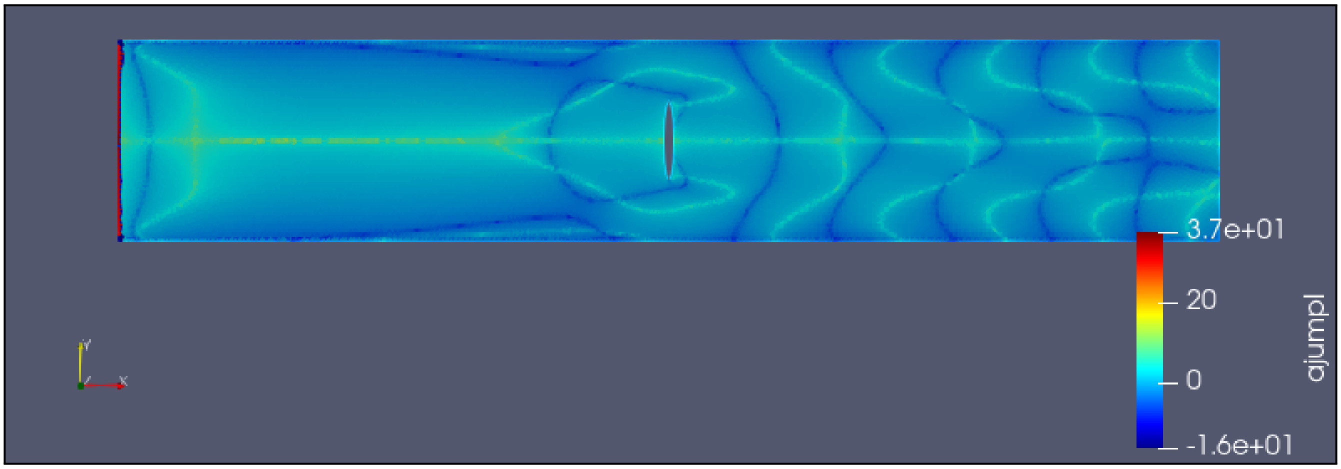


**Figure 45.** Upstream waves - ajumpL

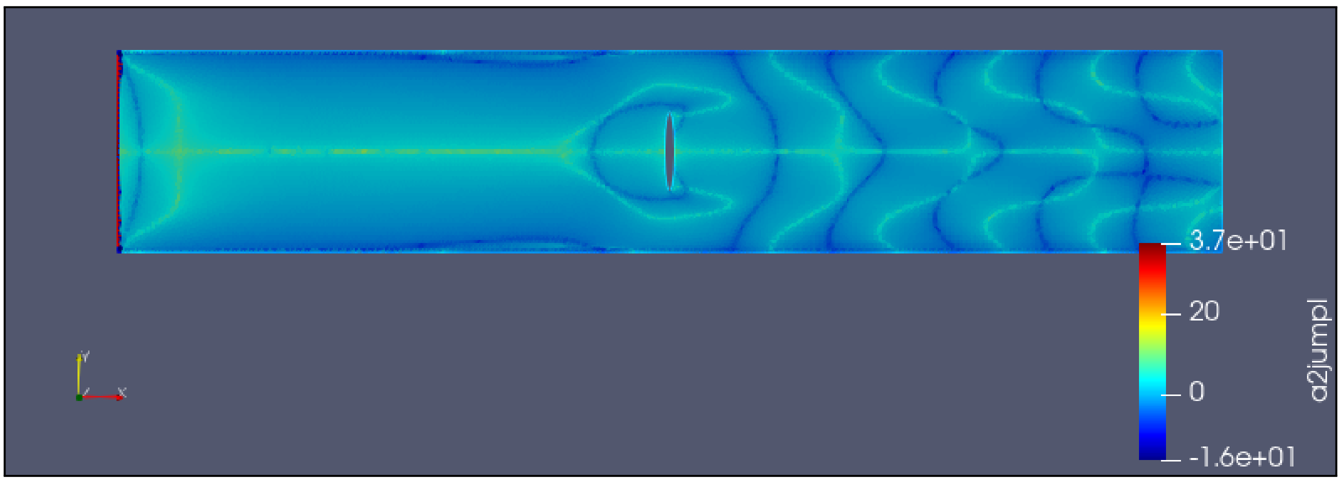


**Figure 46.** Upstream waves - a2jumpL

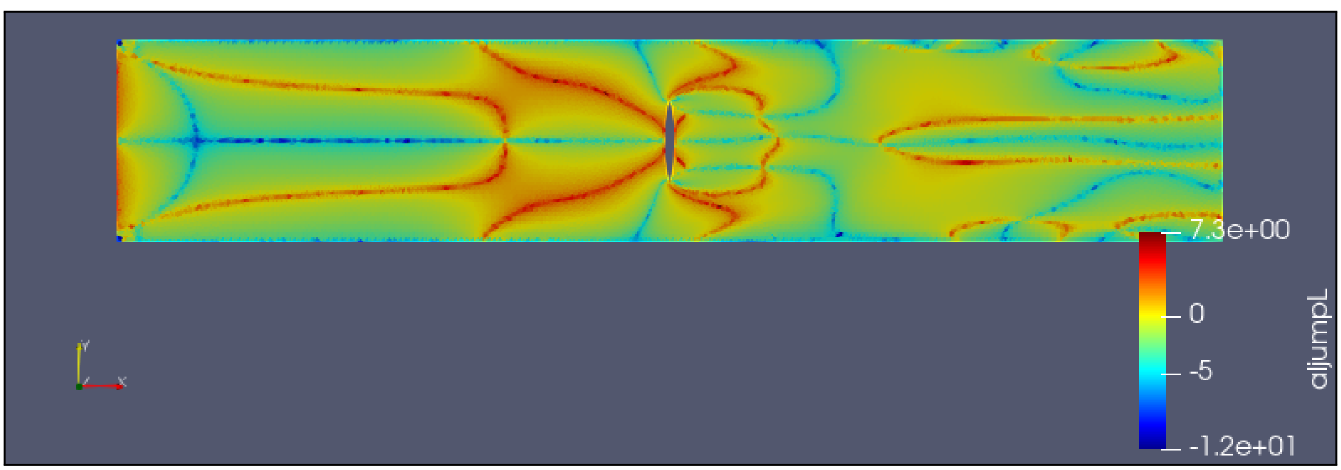


**Figure 47.** aljumpL

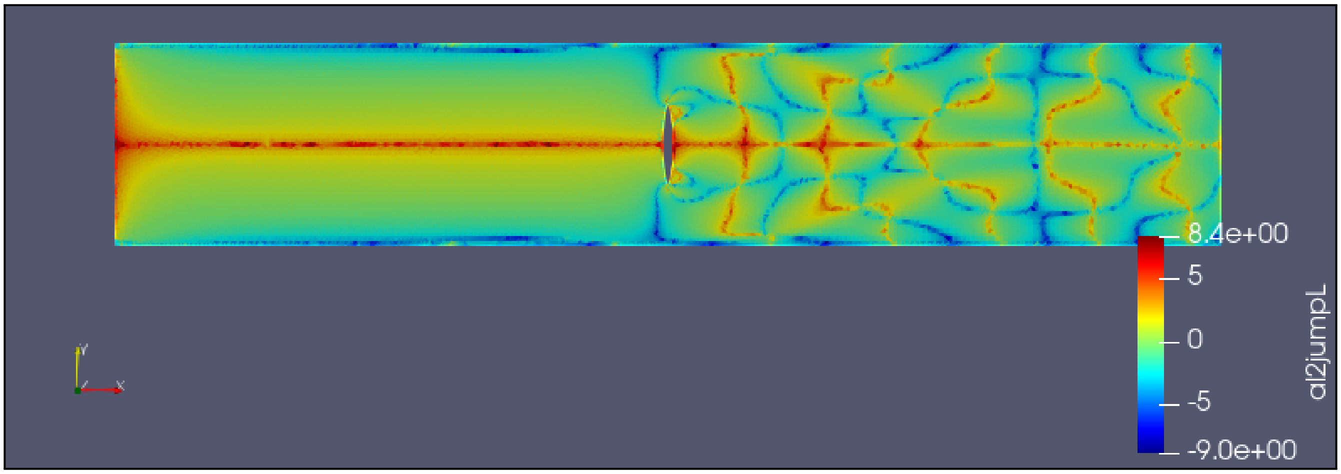


**Figure 48.** al2jumpL - sequence 1

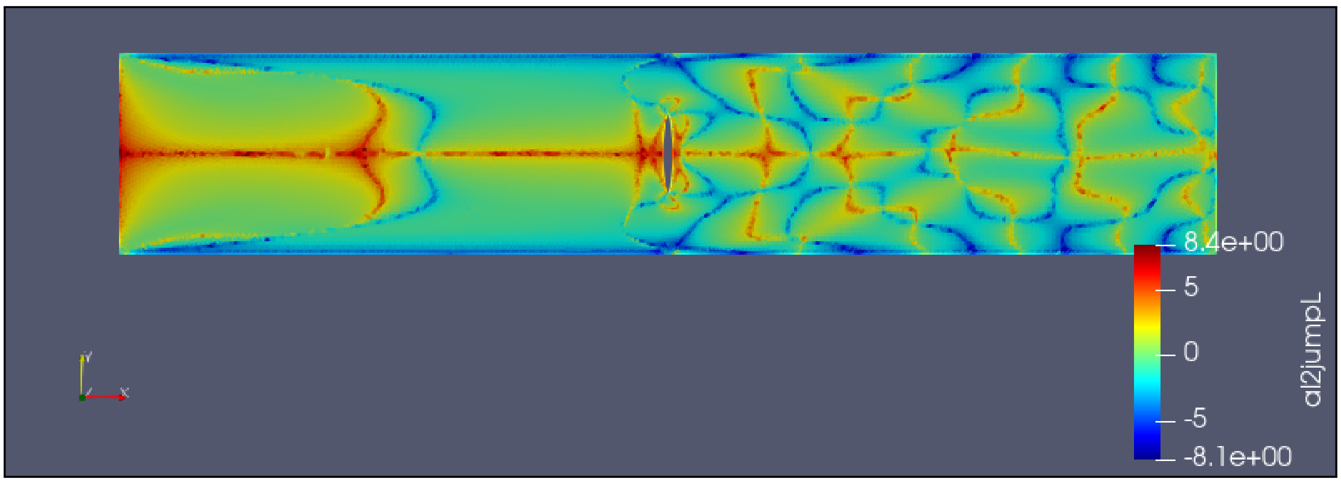


**Figure 49.** al2jumpL - sequence 2

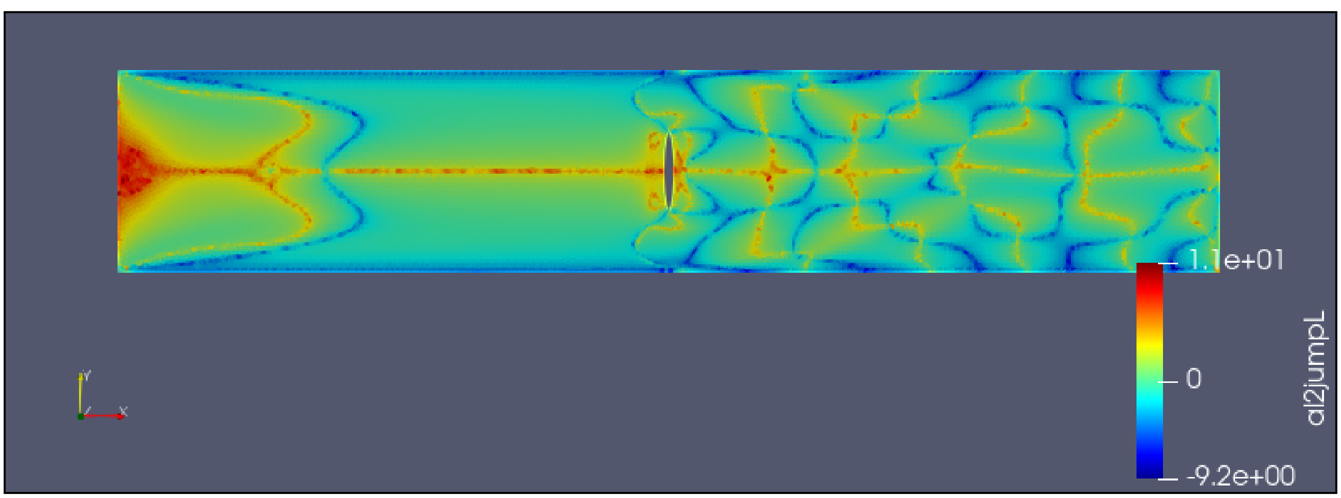


**Figure 50.** al2jumpL - sequence 3

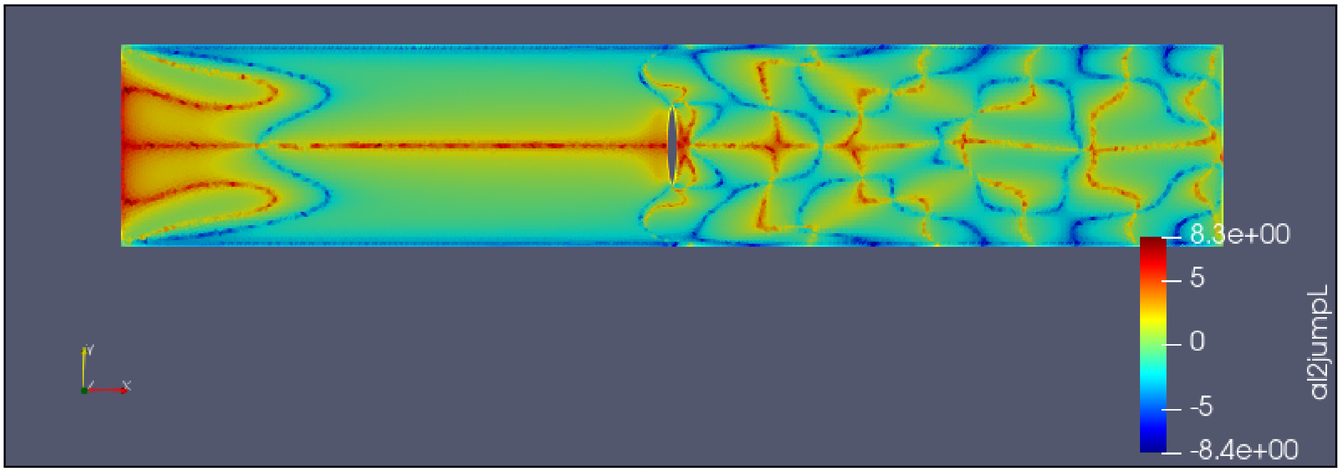


**Figure 51.** al2jumpL - sequence 4

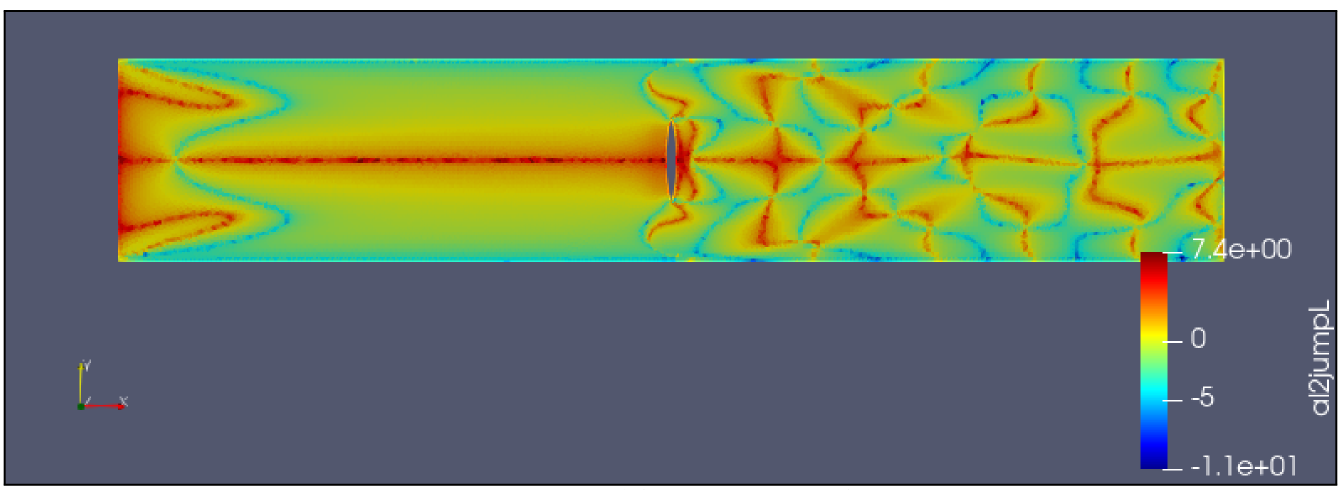


**Figure 52.** al2jumpL - sequence 5