Evolution of FVM model for numerical simulation of compressible flows

M.V. Kraposhin, S.V. Strijhak

ISP RAS, Russia, Moscow

04.11.2015

Contents

- Introduction
- Motivation for development of new solver in OpenFOAM
- Test cases and validation
- First results with different applications
- Future work
- Some conclusions

Different solvers in OpenFOAM

$FOAM_APP/solvers

DNS
Прямое численное моделирование

basic
Простейшие уравнения

combustion
Задачи с горением

compressible
Сжимаемые задачи (в т.ч. с M=1 и M>1)

discreteMethods
Дискретные методы

electromagnetics
Гидро- электромагнетизм

financial
Экономические

heatTransfer
Тепло- и массо- обмен

incompressible
Несжимаемые течения

lagrangian
Течение жидкости с учетом движения отдельных частиц

multiphase
Многофазные течения

stressAnalysis
Анализ прочности

Solver — numerical model for integration of partial differential equations, which is based on finite volume method

Two different versions of OpenFOAM: ***www.openfoam.org*** *and* ***www.extend-project.de***

Development of a new solver pisoCentralFoam for simulation of compressible flows

Motivation:

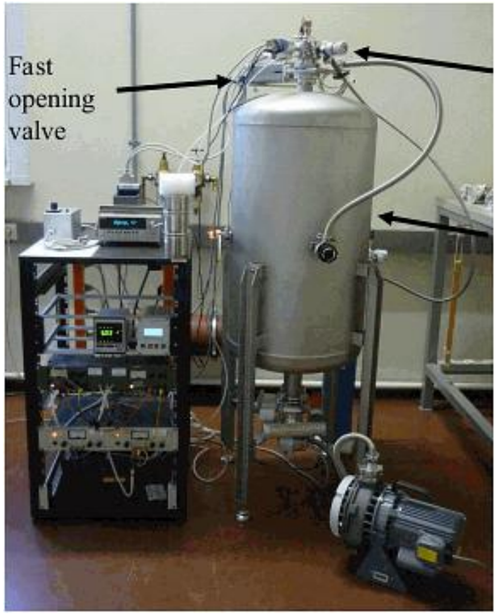
- Simulation of compressible turbulent jets with different M, Re, n numbers by one solver
- Simulation of strong shocks with low dissipation
- Support of models for break-up, evaporation, collision, heat transfer for liquid droplets
- Support of models of combustion for liquid and gas fuels
- Simulation of jets in vacuum and with moving bodies
- Application to nozzles, water and fuel injectors
- Selection of test cases for code's verification
- Development in OpenFoam 2.3, 2.4, foam-extend 3.1

- Applications of rarefied gas flows:

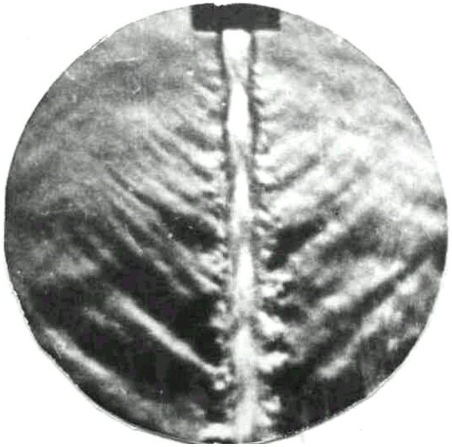
  - Micro-devices ; High-altitude aerodynamics

  - Vacuum technology (10-100 Pa); Physical/Chemical vapor deposition;

  - Cryogenics ; Scientific experiments

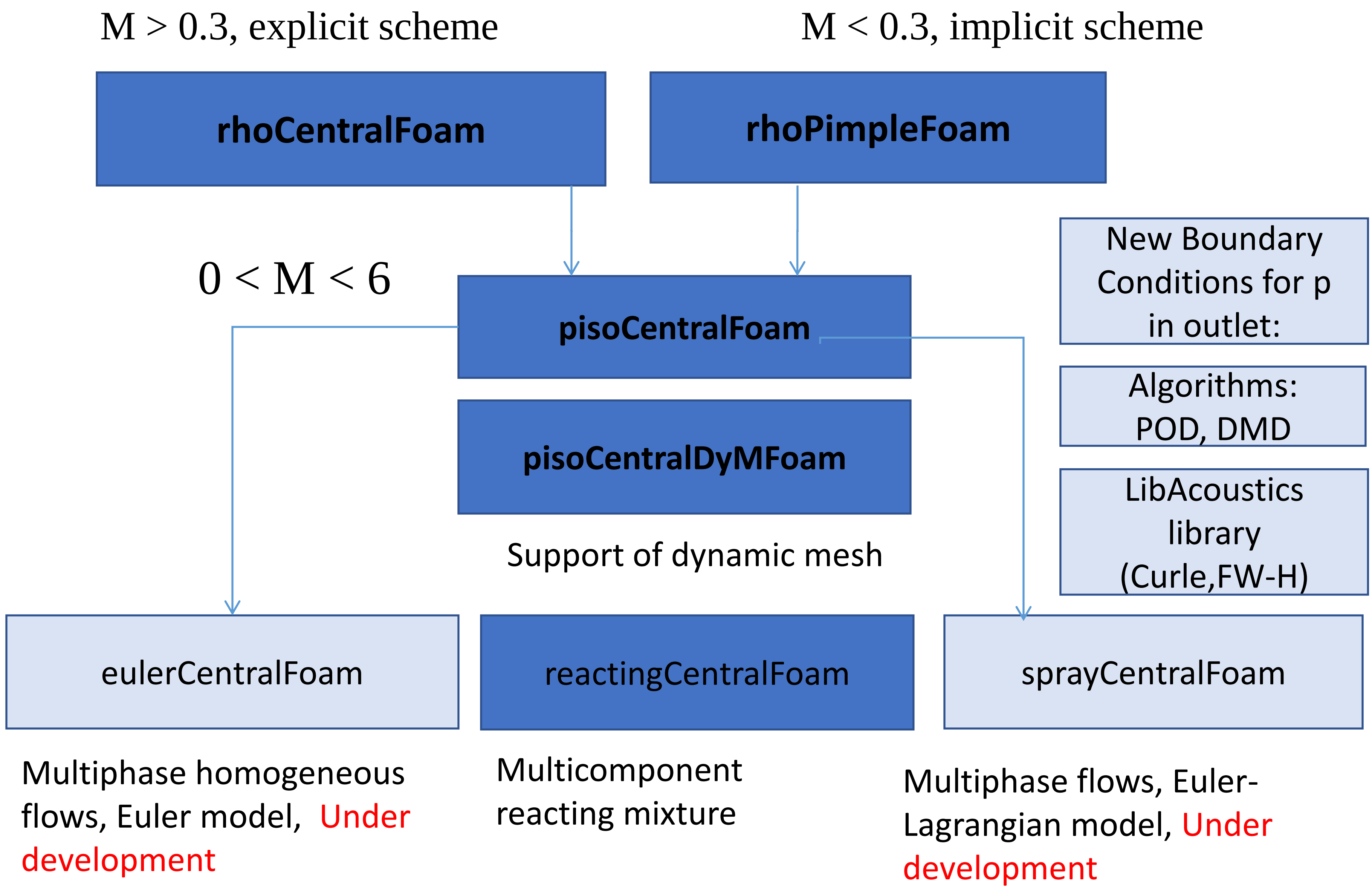


Vacuum camera

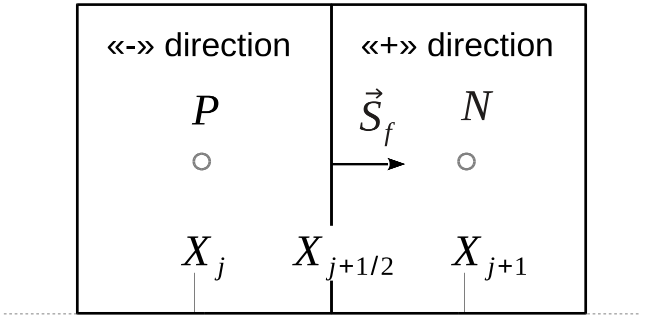


Jet impinging on plate

Solver with hybrid scheme for different Mach numbers



ESSENTIALS OF HYBRID KT/PISO SCHEME



$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{U}) = 0$$

$$\frac{\partial \rho \vec{U}}{\partial t} + \nabla \cdot (\rho \vec{U} \vec{U}) = \nabla \cdot \Pi + \vec{F_b}$$

$$\frac{\partial \rho e}{\partial t} + \nabla \cdot (\rho \vec{U} e) = \nabla \cdot (\Pi \cdot \vec{U}) + \vec{F_b} \cdot \vec{U} - \nabla \cdot \vec{q}$$

Standard FVM: Segregated approach + PISO $\int_V \nabla \cdot (\vec{U}\Psi)\, dV = \int_S d\vec{S} \cdot (\vec{U}\Psi) = \sum_f \Psi_f (\vec{U} \cdot \vec{S}_f) = \sum_f \Psi_f \varphi_f$

Where $\Psi_f \phi_f$ is linear interpolation from volumes to faces

Kurganov-Tadmore scheme: interpolate data using upwind-central scheme and maximum sound propagation speeds in adjacent cells

$$\Psi_f \varphi_f = \Psi_f^P (\alpha_f^P \varphi_f^P + \alpha_f^P a_f^{min}) + \Psi_f^N (\alpha_f^N \varphi_f^N - \alpha_f^P a_f^{min})$$

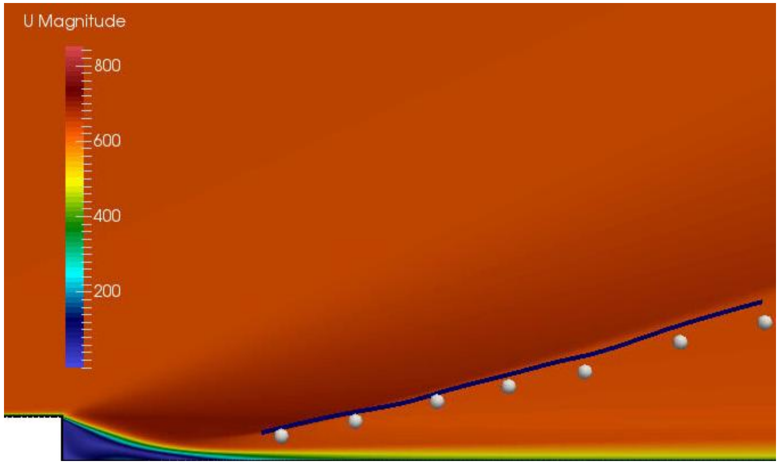
Hybrid scheme: mix Kurganov-Tadmore scheme with standard linear/upwind using blending factor κ_f

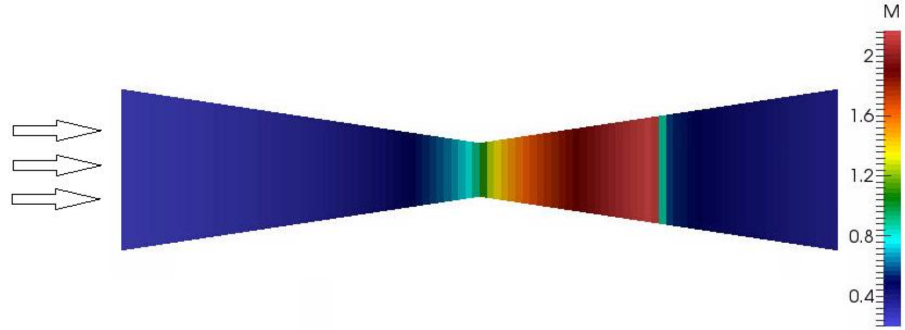
$$\Phi_f^P = \kappa_f \rho_f^P (\alpha_f^P \varphi_f^P + \alpha_f^P a_f^{min})$$

$$\Phi_f^N = (1 - \kappa_f) \rho_f^P (\alpha_f^P \varphi_f^P + \alpha_f^P a_f^{min}) + \rho_f^N (\alpha_f^N \varphi_f^N - \alpha_f^P a_f^{min})$$

$$\kappa_f = min\left({M_f}/{CFL}, 1 \right)$$

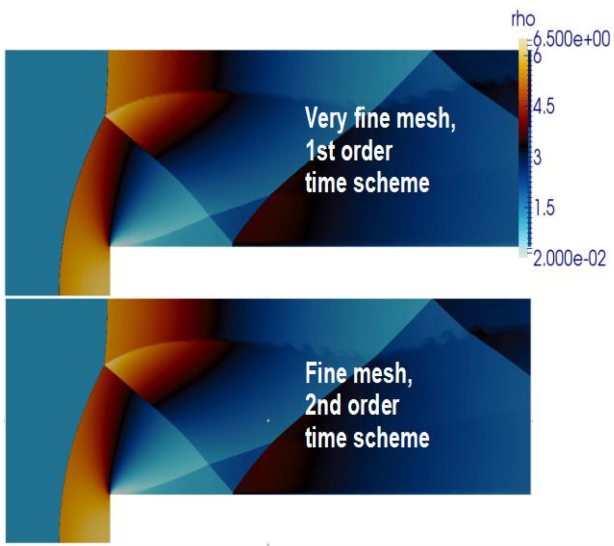
3 test cases for validation



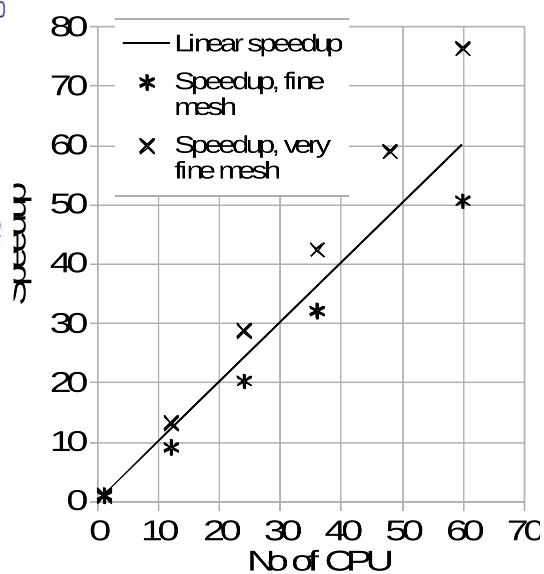


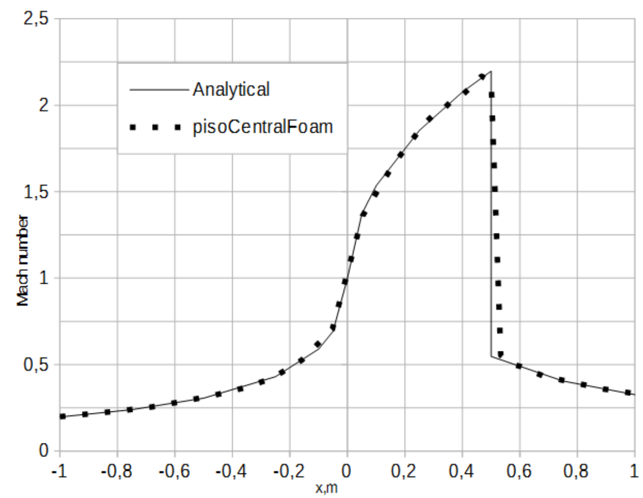
Backward facing step. URANS + k-omega SST.

Flow in a supersonic nozzle



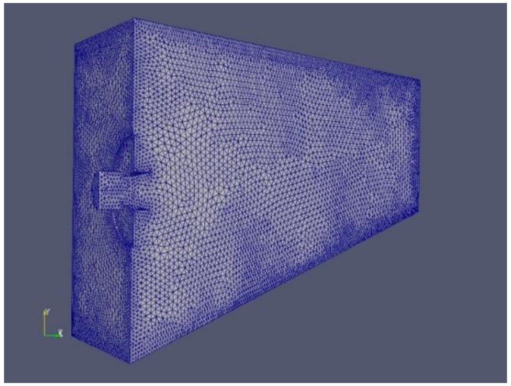
Forward-facing step problem

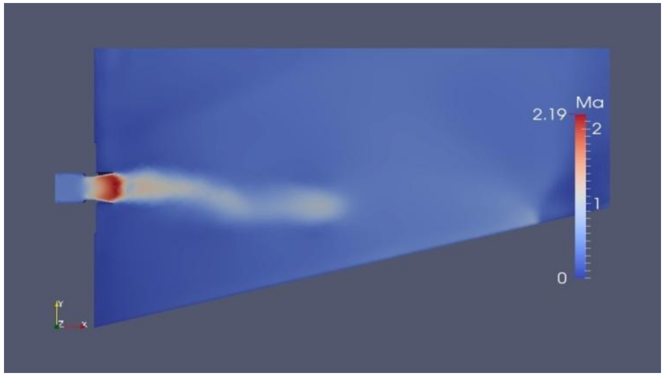


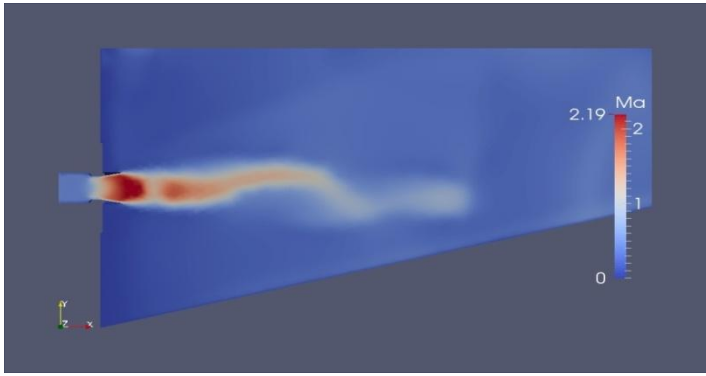


Flow in a supersonic nozzle

Simulation of compressible turbulent jets

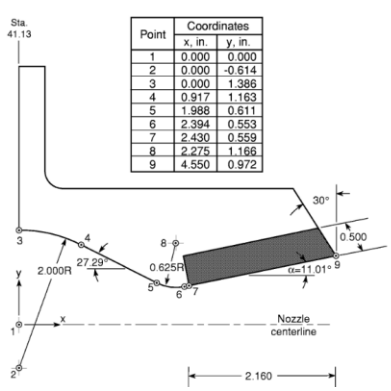


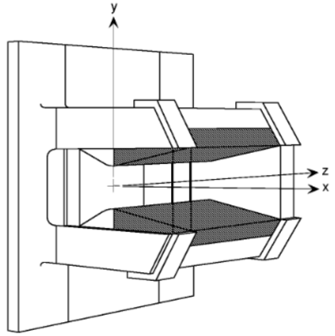




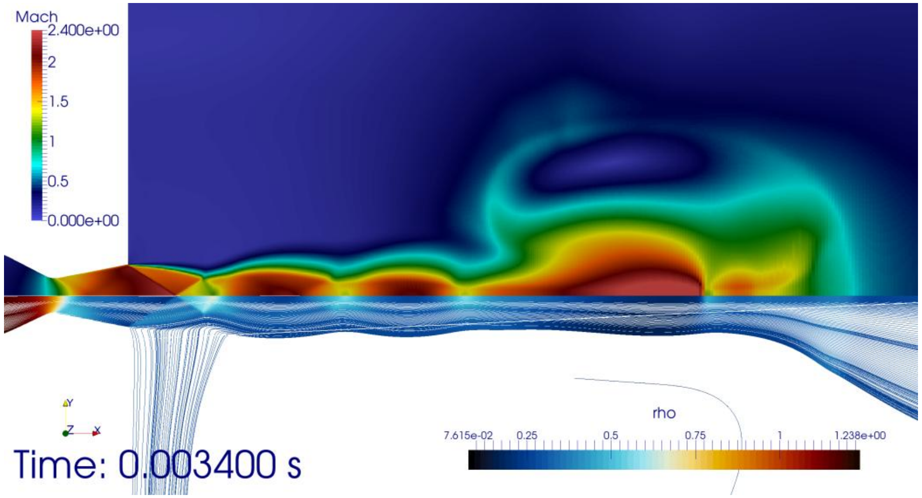
Nozzle and tank with inclined wall Mach field number at different times with pisoCentralFoam

Geometry and mesh were built with Salome. Mj=2.2, n=2.





Nozzle with porous wall. NASA experiment



Experiment of ITAM (http://www.itam.nsc.ru/)
V.I. Zapryagaev et al. Journal of Applied Mechanics and Technical Physics . 2011

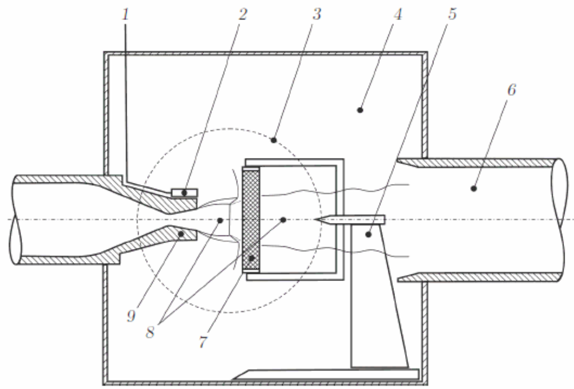


Рис. 1. Схема экспериментальной установки для исследования струйных потоков:
1 — выход на цифровую систему сбора данных; *2* — датчик пульсаций давления; *3* — смотровое окно; *4* — камера Эйфеля; *5* — пилон, перемещаемый вдоль оси струи; *6* — диффузор; *7* — сменная преграда; *8* — струя (сверхзвуковая недорасширенная перед преградой и дозвуковая за преградой); *9* — сопло

Experimental test camera for jets

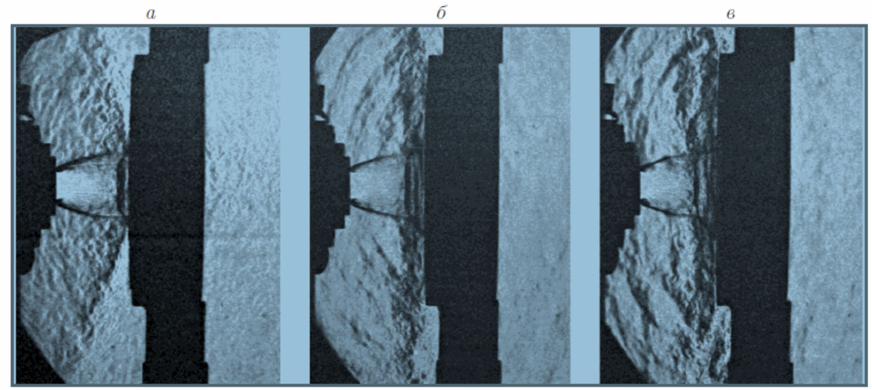


Рис. 3. Фотографии струи, натекающей на различные преграды:
а — пористая преграда R1; *б* — пористая преграда R2; *в* — непроницаемая преграда R3

Jet impinging on plate with solid and porous walls

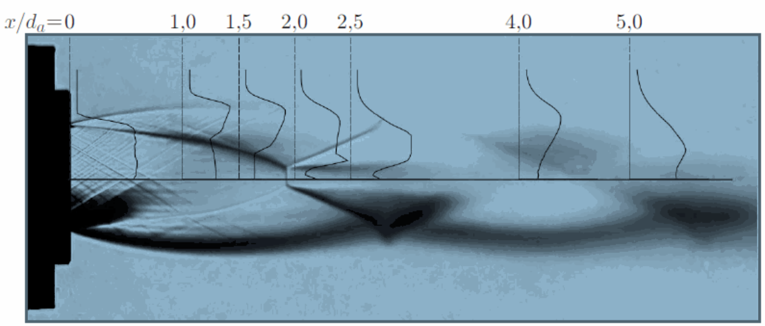
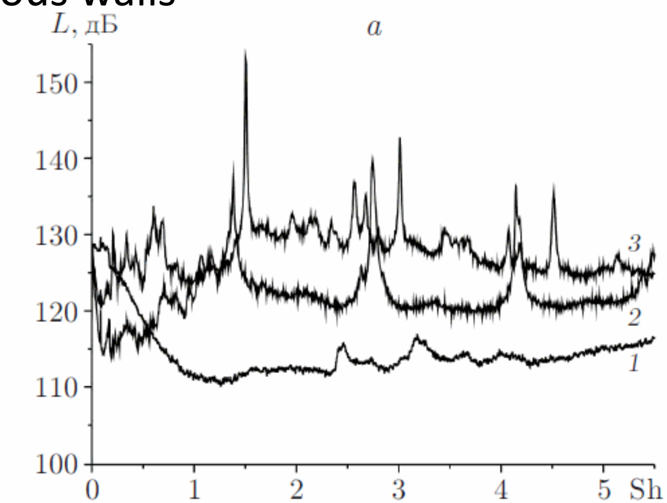


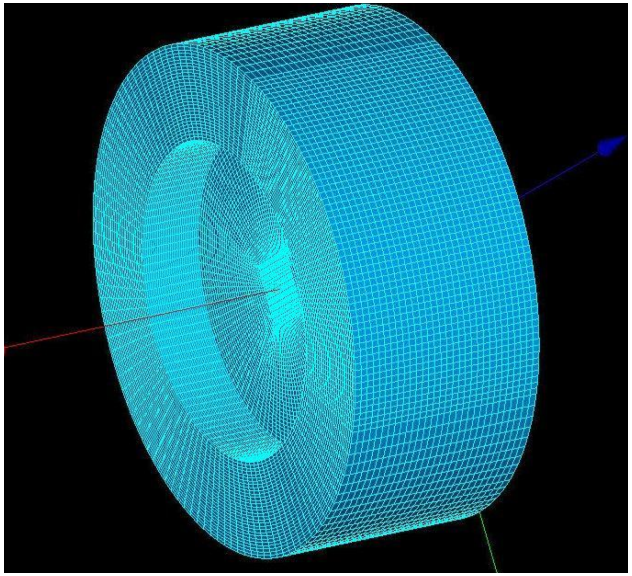
Рис. 2. Шлирен-фотография сверхзвуковой недорасширенной струи ($M_a = 1{,}72$, $n = 2{,}1$)

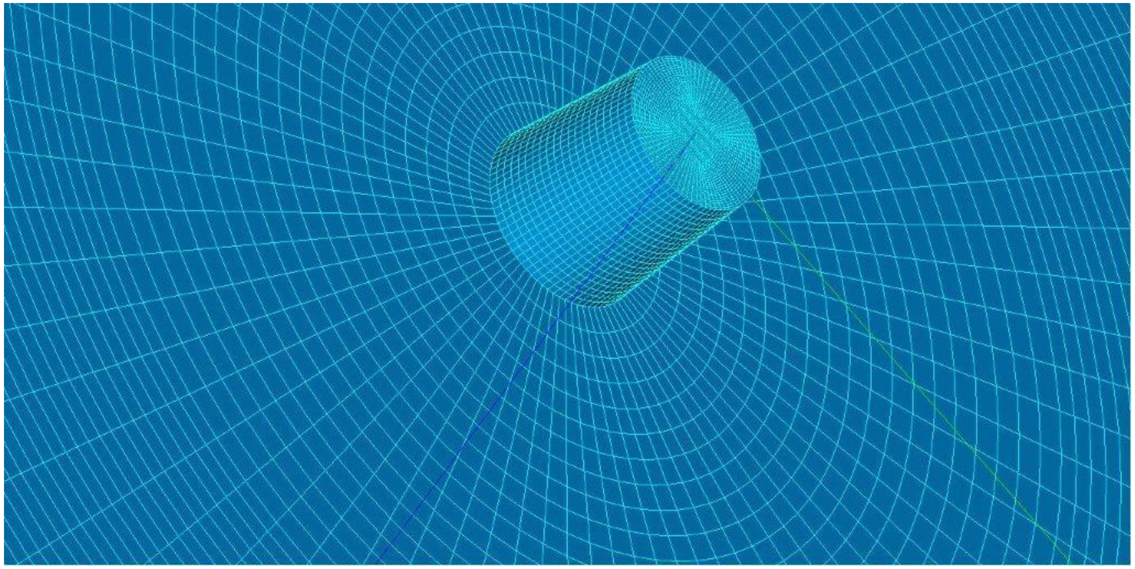
Schlieren photography Ma=1.72, n=2.1

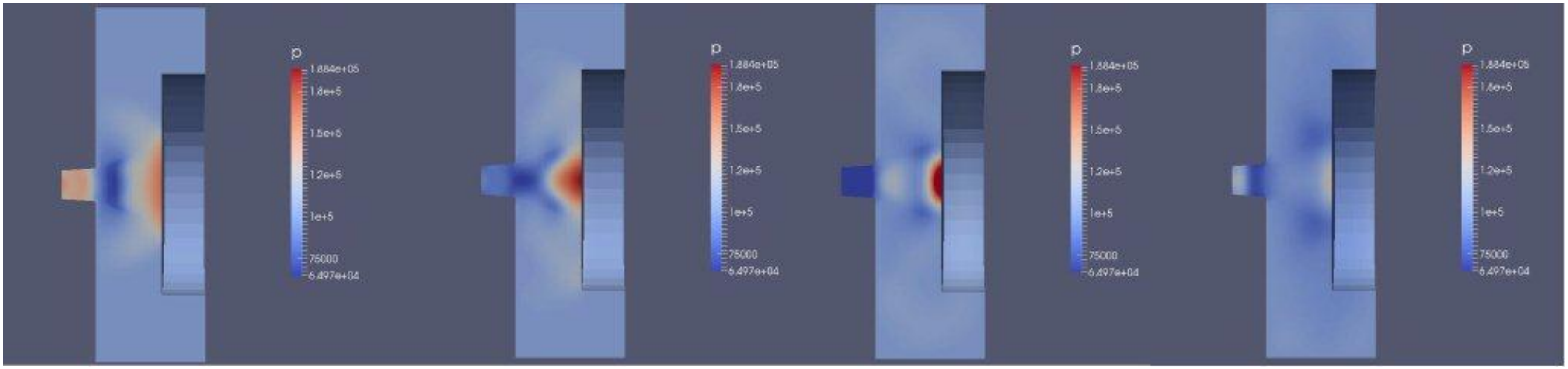


Sound Pressure Level at x/da = 1,0.

Simulation of impinging jet on plate with pisoCentralFoam

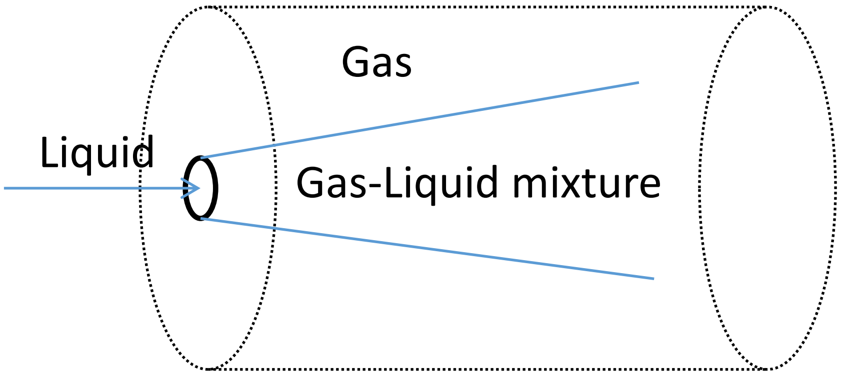


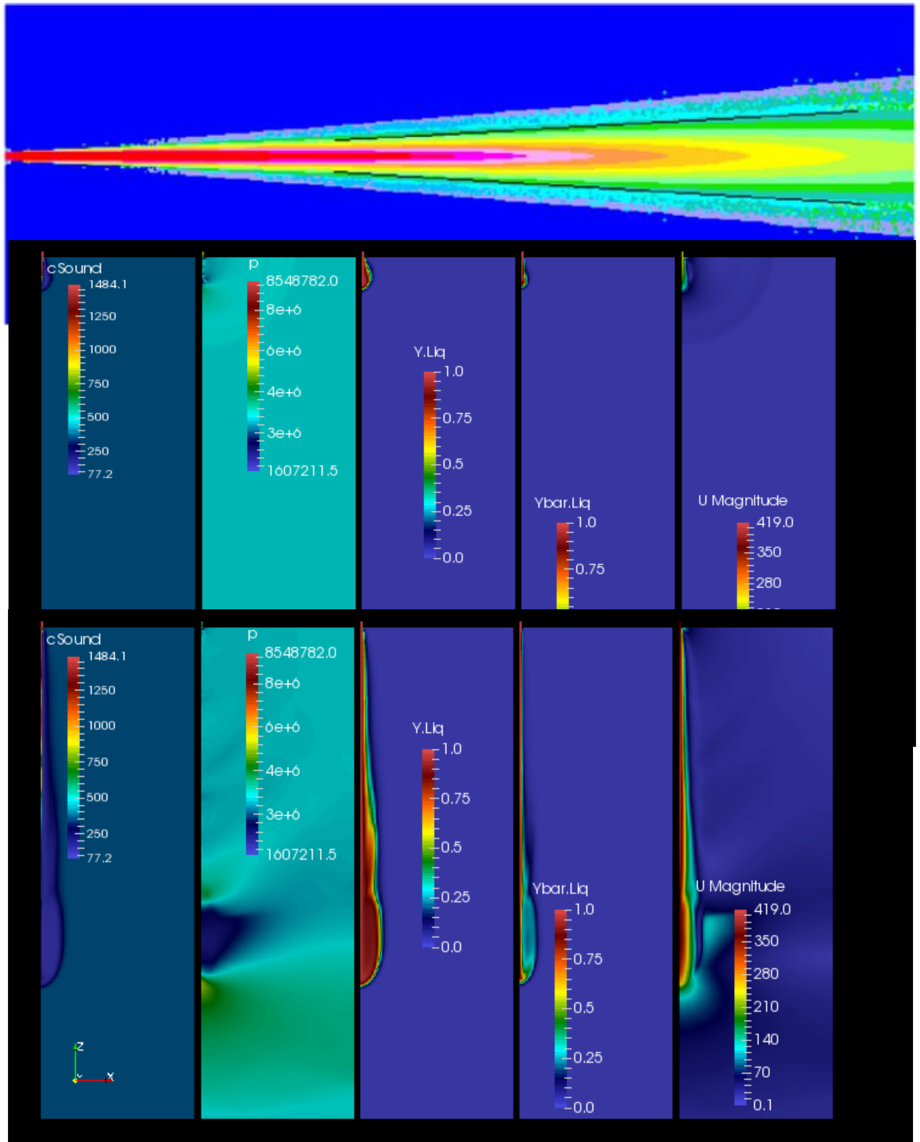




Multi-block grid with Salome and simulation were done by our colleague Arina Bovtrikova

APPLICATION OF HYBRID KT/PISO SCHEME TO THE FLOW OF COMPRESSIBLE TWO PHASE MIXTURE





$$\frac{\partial \rho}{\partial t} + \nabla \bullet \left(\rho \vec{U}\right) = 0$$

$$\frac{\partial \rho \vec{U}}{\partial t} + \nabla \bullet \left(\rho \vec{U} \otimes \vec{U}\right) = -\nabla p + \nabla \bullet \sigma + \rho \vec{g}$$

$$\frac{\partial \rho h}{\partial t} + \nabla \bullet \left(\rho \vec{U} h\right) = -\frac{\partial \rho K}{\partial t} - \nabla \bullet \left(\rho \vec{U} K\right) + \frac{\partial p}{\partial t} + \nabla \bullet \sigma + \rho \vec{g} \bullet \vec{U} + \nabla \bullet \lambda \nabla T$$

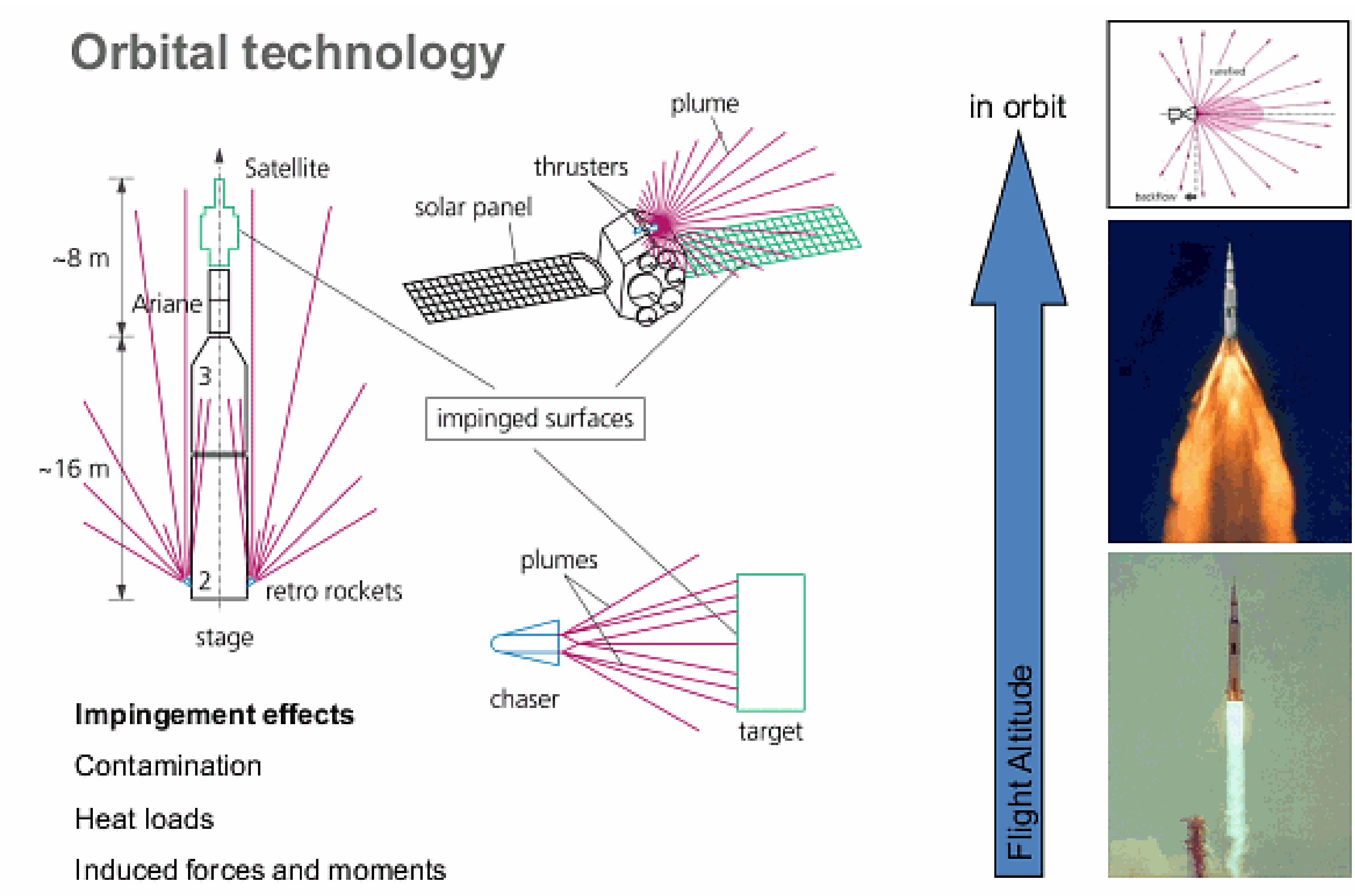
$$\frac{\partial \rho Y_{Liq}}{\partial t} + \nabla \bullet \left(\rho \vec{U} Y_{Liq}\right) = \nabla \bullet \left(\vec{U}_r \rho Y_{Liq}\right)$$

$$1/\rho = Y_{Liq}/\rho_{Liq} + Y_{Gas}/\rho_{Gas}$$

$$Y_{Liq} + Y_{Gas} = 1$$

$$\bar{Y}_{Liq} = \frac{\rho}{\rho_{Liq}} Y_{Liq}$$

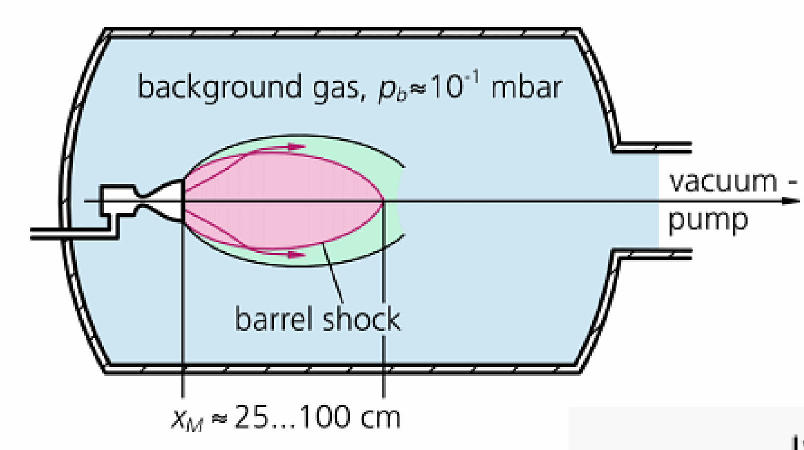
Different research topics in aerospace industry

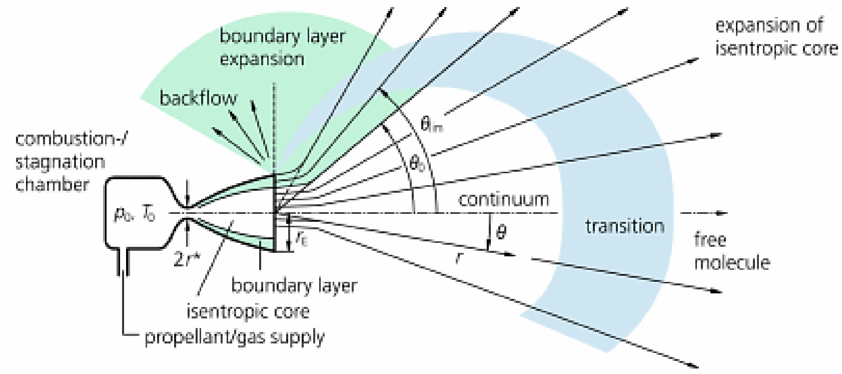


Classification of flow regimes

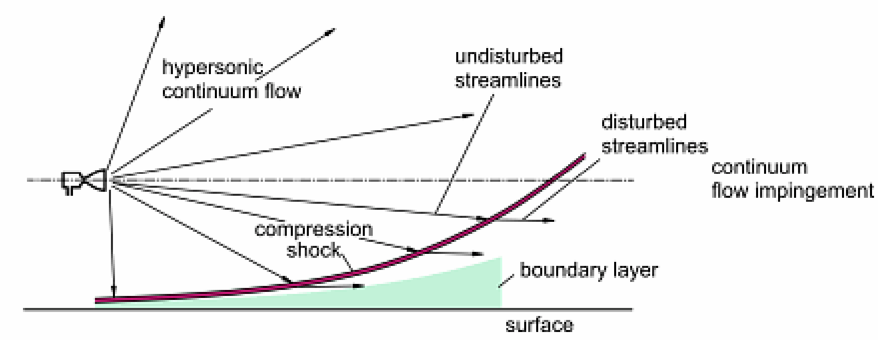
Kn values	Regime	Mathematical formulation	Numerical method
Kn<10e-3	Hydrodynamic Continua media approach	Navier-Stokes equations	Classical numerical schemes
10e-3<Kn<10e-1	Slip	Navier-Stokes equations with slip boundary conditions	Classical numerical schemes
10e-1<Kn<100	Transit	Boltzmann equation, Kinetic models	DVM (a discrete velocity model), DSMC, Integral methods
Kn>100	Free molecules	Collision less Boltzmann equation	Characteristics method TPMC (test particle Monte Carlo)

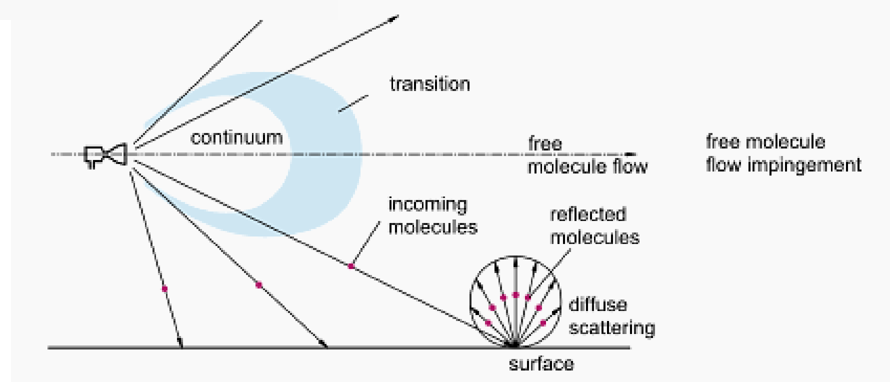
Different models of jets in vacuum camera for space engine





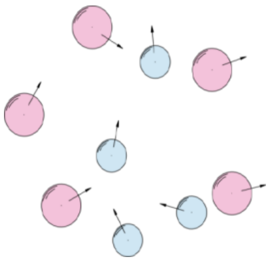
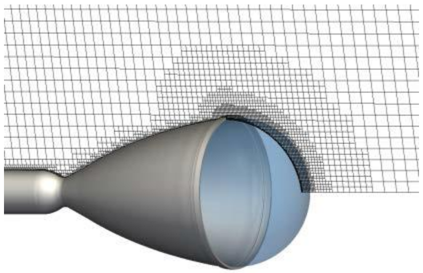
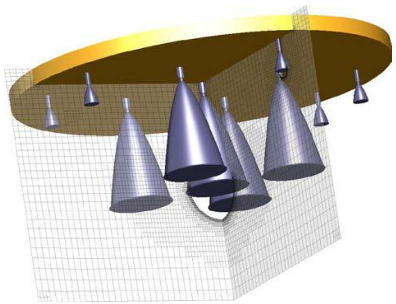
$$\mathrm{Kn}_Q = \lambda \frac{|\nabla Q|}{Q}, \quad Q \in \{\rho, u, T\}.$$





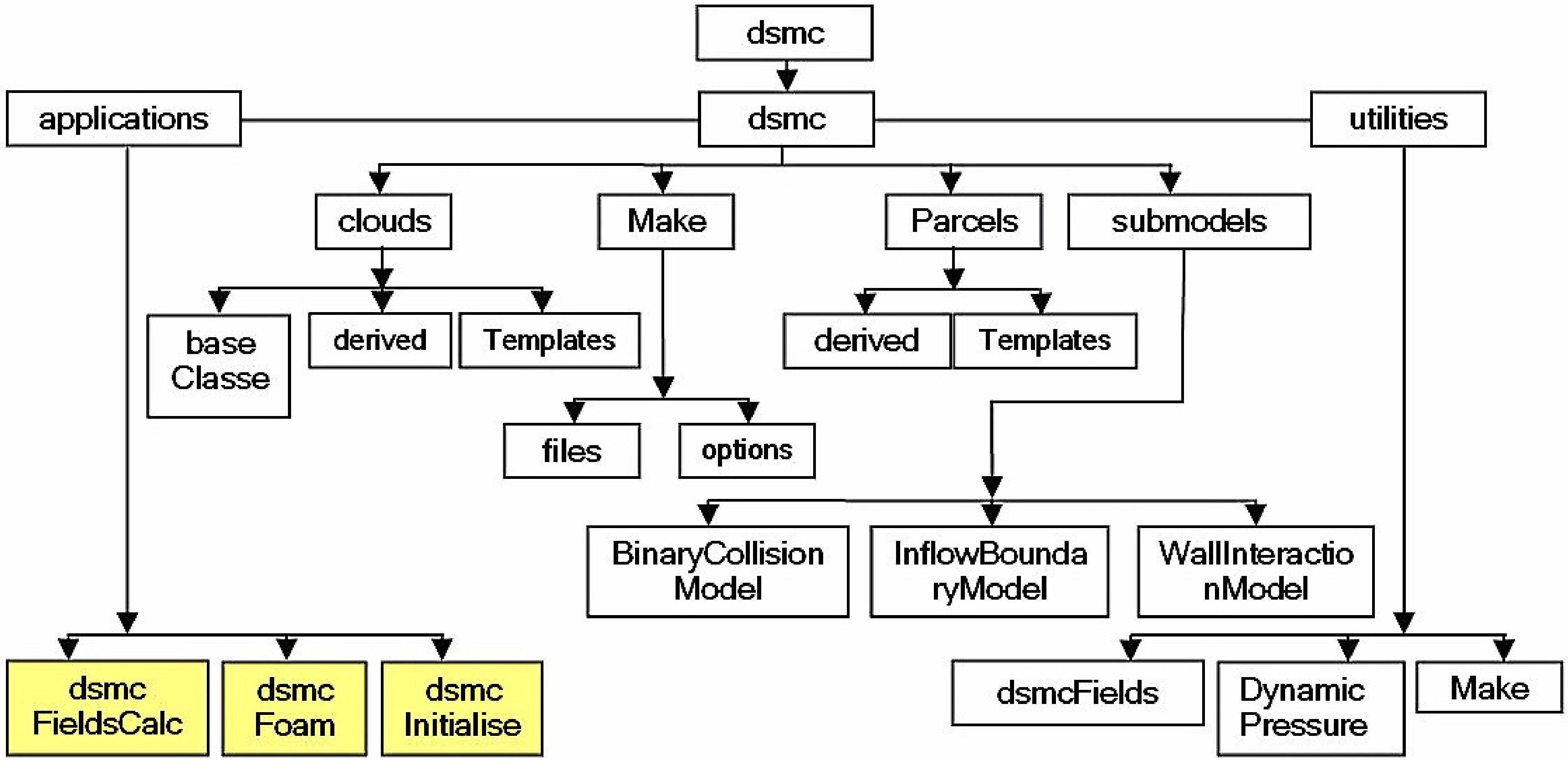
G.Dettleff , M. Grabe Basics of Plume Impingement Analysis for Small Chemical and Cold Gas Thrusters. DLR. RTO-EN-AVT-194. 2011. p.40.

dsmcFoam – Direct Simulation Monte Carlo solver in OpenFOAM

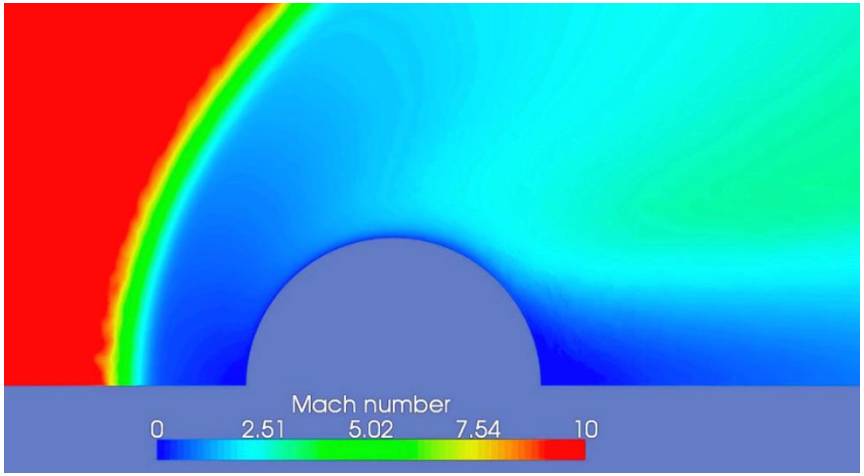
  

- Need for modeling of rarefied flows at high altitude (h~ 100 km) and low P
- **DSMC (Direct Simulation Monte Carlo Method)** – the stochastic method of particles based on the kinetic theory of gases.
- **Kinetic theory** – the statistical theory of dynamics of systems with a large number of molecules. Describes macroscopic parameters of system in terms of microscopic motion of the molecules and their interaction, uses probability theory.
- The calculation of particles represents calculation of real molecules. Their position, velocity, energy are calculated with time.
- **dsmcFoam** – developed by OpenCFD together with University of Strathclyde, Scotland, Glasgow.
- **dsmcFoamStrath** – new solver which includes multicomponent mixtures with chemical reactions.

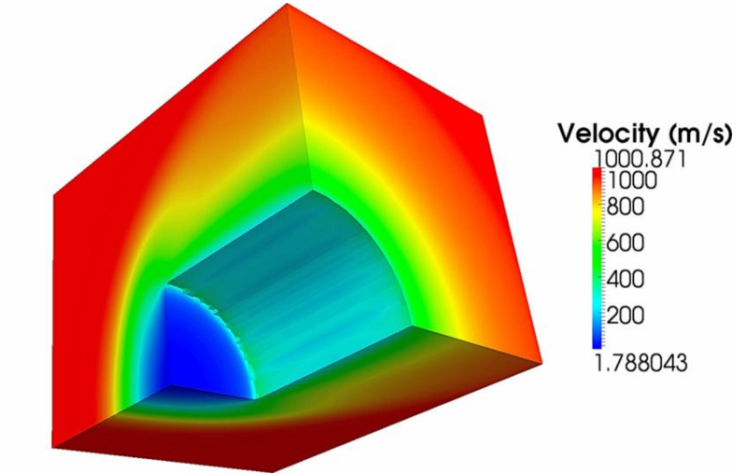
Solver dsmcFoam (module's structure)



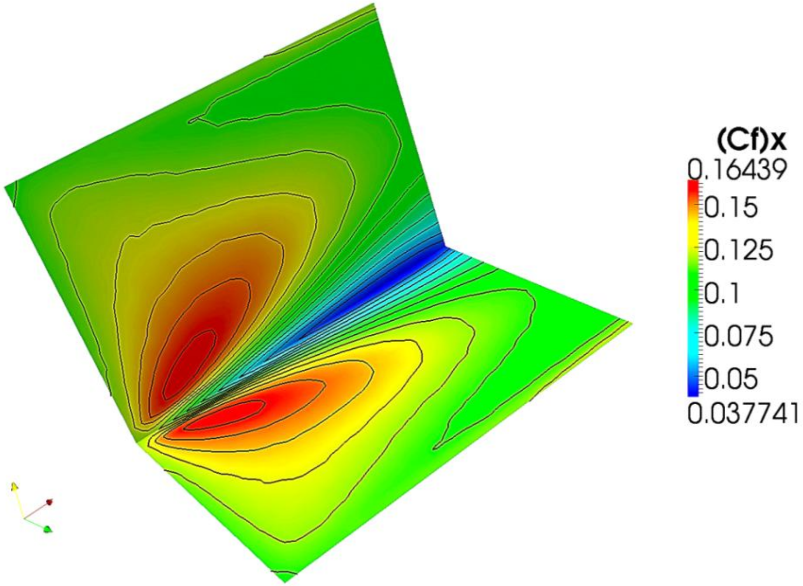
Test cases with dsmcFoam



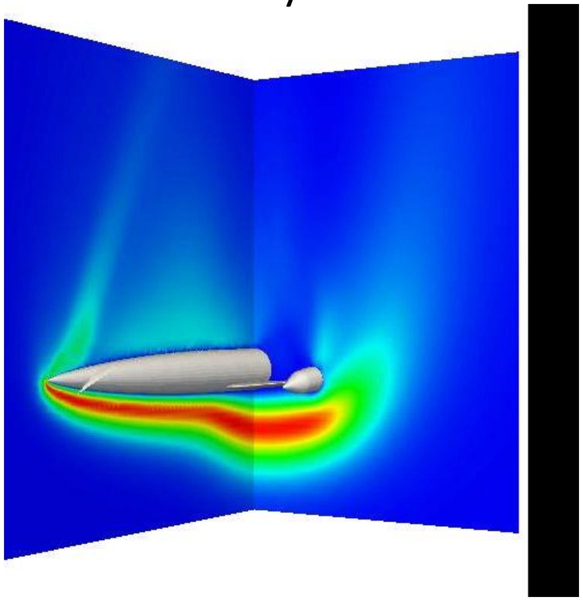
2D Cylinder



3D Cylinder

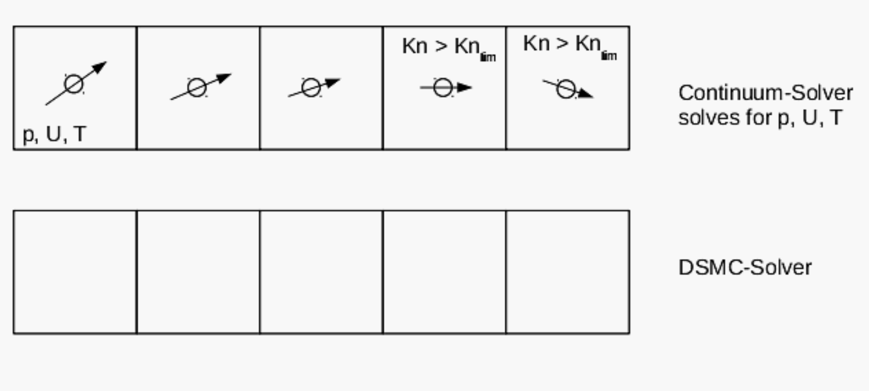


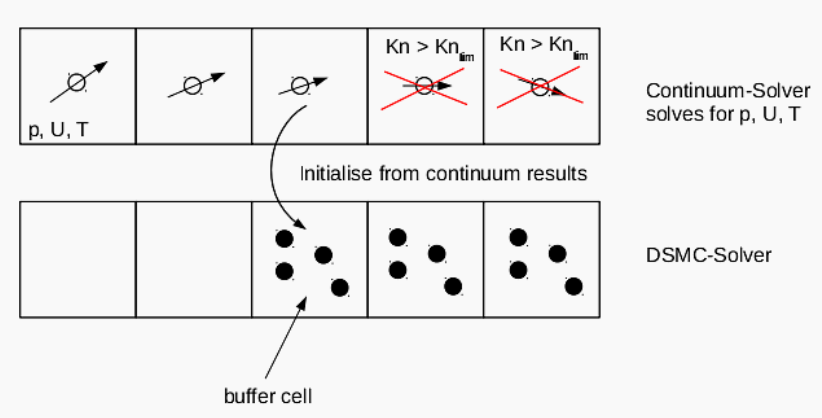
3D Corner



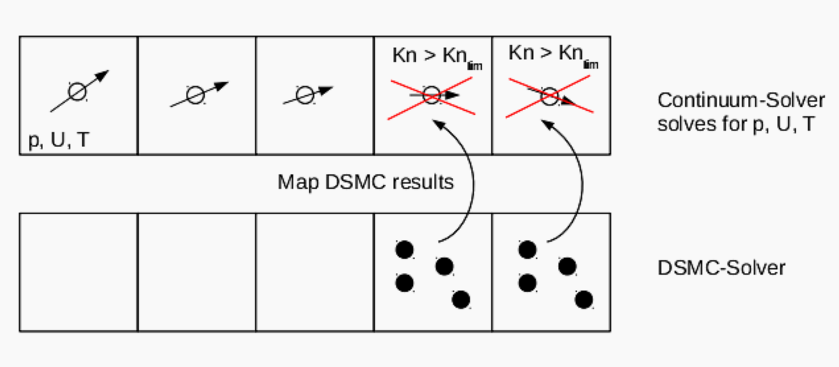
M=21 H=114 км

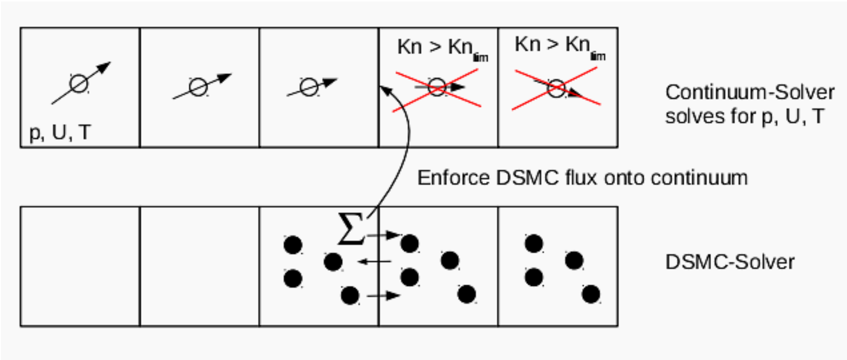
Main idea of algorithm for hybrid continuum-particle solver for unsteady rarefied gas flows





We need to define local Kn number in each cell

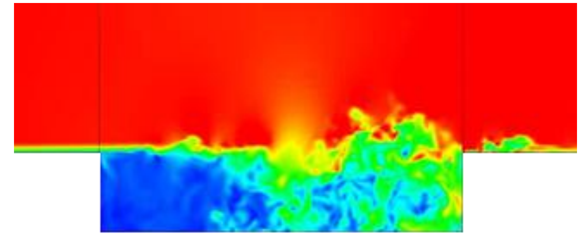




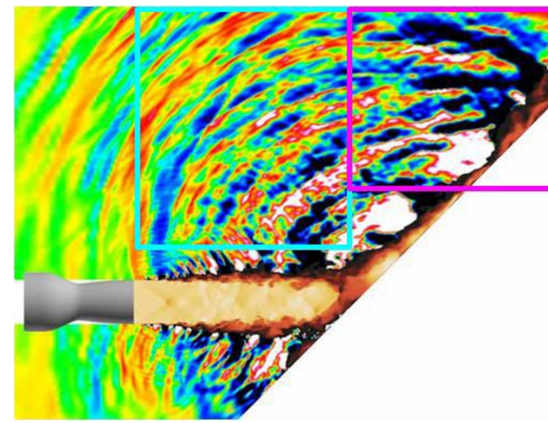
S. Pantazis, H. Rusche, A hybrid continuum-particle solver for unsteady rarefied gas Flows, Vacuum, 2014. DOI: 10.1016/j.vacuum.2014.06.022 (rhopimpleFoam + dsmcFoam)

Motivation

Despite improved algorithms and powerful supercomputers, “high-fidelity” models are often too expensive for use in a design or analysis setting.

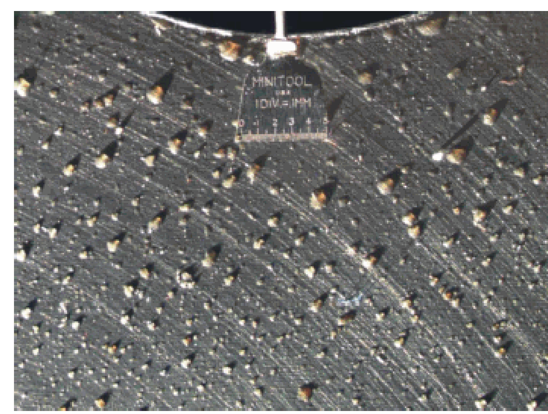


Example of application area in which this situation arises: compressible cavity flow problem, jet impinging on plate.



Large Eddy Simulations with very fine meshes and long times are required to predict accurately dynamic pressures loads, heat fluxes, sedimentation of particles or debris transport on specified element of interest

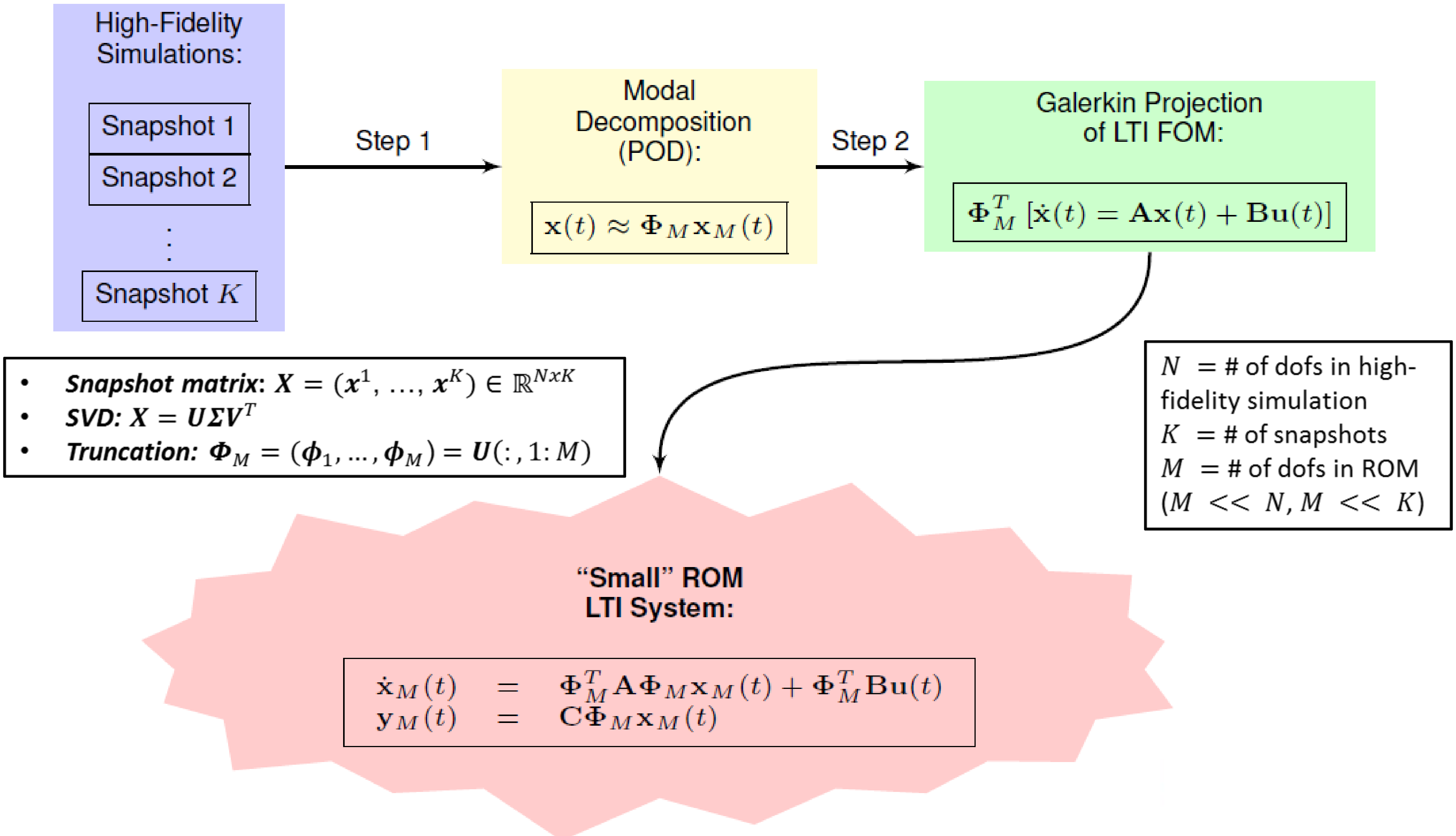
These simulations take ***months or years*** even when run in parallel on state-of-the-art supercomputers!



Droplets of incompletely burnt fuel

Need for simplified models for a long real time!

Proper Orthogonal Decomposition (POD)/ Galerkin Method to Model Reduction



Conclusions

- New solver pisoCentralFoam is developed
- pisoCentralFoam with different modifications is available for public download on **sourceforge.net**
- Our solver is a good start point for scientific collaboration with other different groups!
- We plan to develop a hybrid solver for rarefied flows in 2016
- Plans to develop POD/DMD & ROM in scope of Big Data ISP RAS project
- Thanks for support: A. Avetisyan, O. Samovarov,

  M. Kalugin, E. Khashba, O. Andreev and others. ☺