Droplet formation

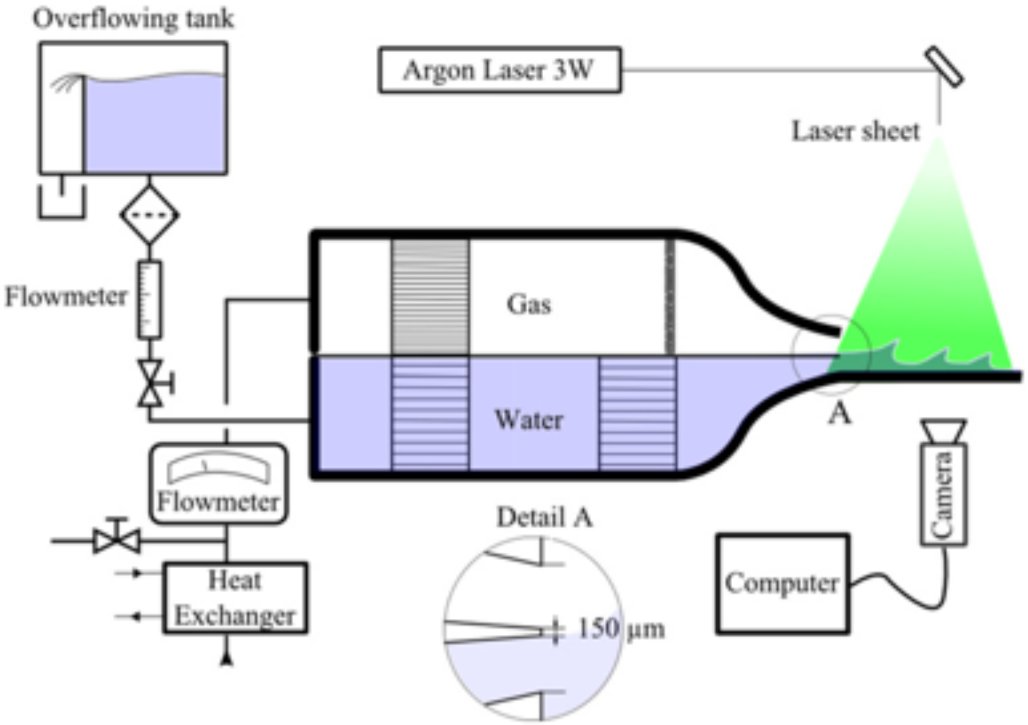
Stéphane Zaleski

Institut Jean Le Rond ∂'Alembert,
CNRS & Université Pierre et Marie Curie -- UPMC – Paris 6

web site http://www.ida.upmc.fr/~zaleski

d'alembert

Grenoble experiment



Descamps et al, 2008
Matas et al., 2011
Jérôme et al, 2013
Fuster et al, 2013
Ling et al, 2015

and of course Hopfinger, Lasheras, Cartellier, Villermaux, Hoepffner, Popinet, Boeck, Rossi ...

1) Is it possible to do a real Direct Numerical Simulation of atomisation, resolving all the scales ?

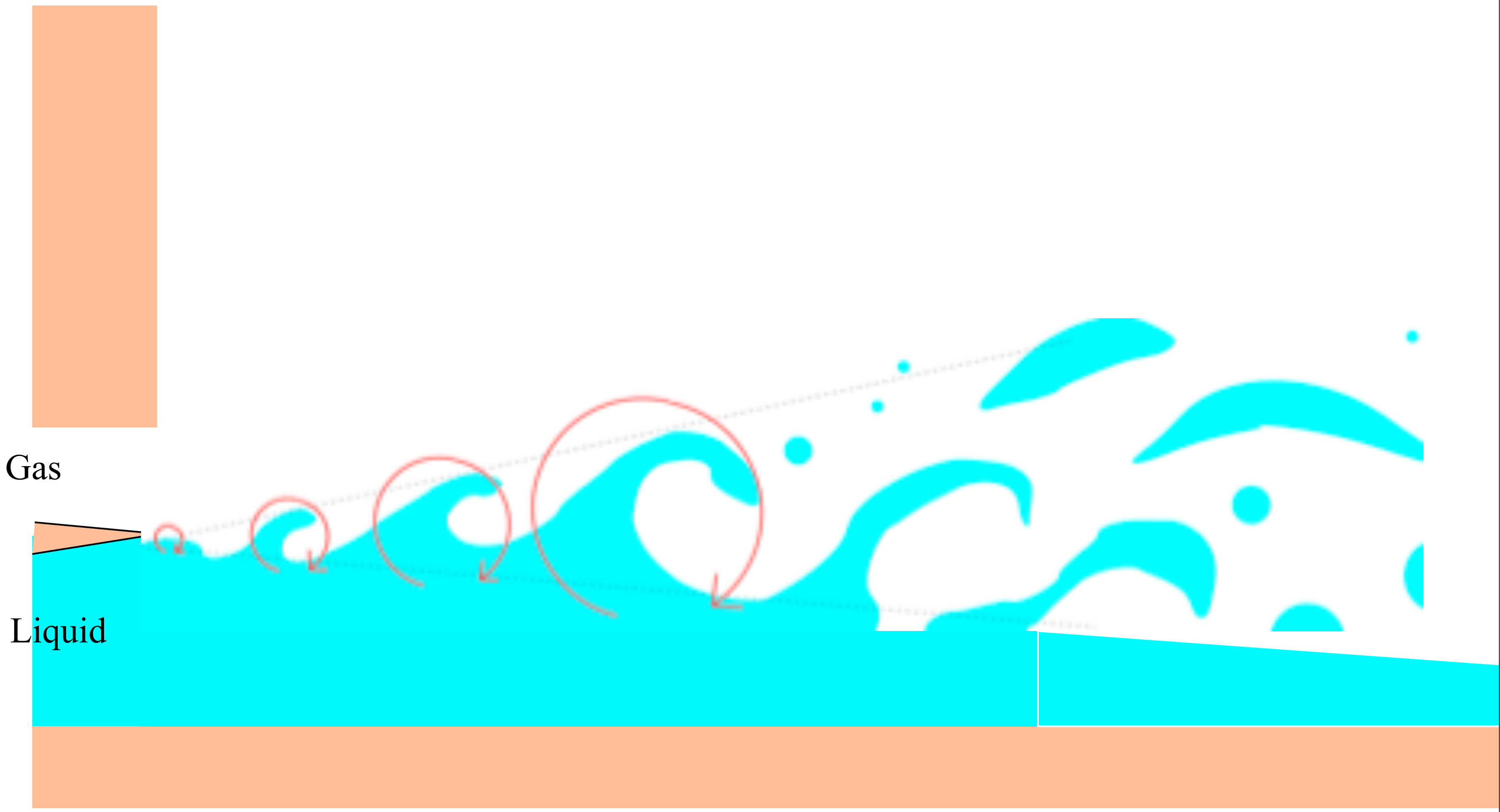
2) What can we learn from these very detailed simulations ?

1) 2D flows

2) 3D flows

2D simulations of the planar « Grenoble » setup.

The Grenoble quasi 2D experiment set up



Navier-Stokes equations with interfaces

$$\partial_t(\rho \mathbf{u}) + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) = -\nabla p + \nabla \cdot (2\mu \mathbf{D}) + \sigma\kappa\delta_s \mathbf{n} + \rho \mathbf{g},$$

where the strain-rate tensor is:

$$D_{ij} = \frac{1}{2}\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right),$$

and both fluids are considered incompressible

$$\nabla \cdot \mathbf{u} = 0\,.$$

Compressible fluids: possible but difficult and less relevant.

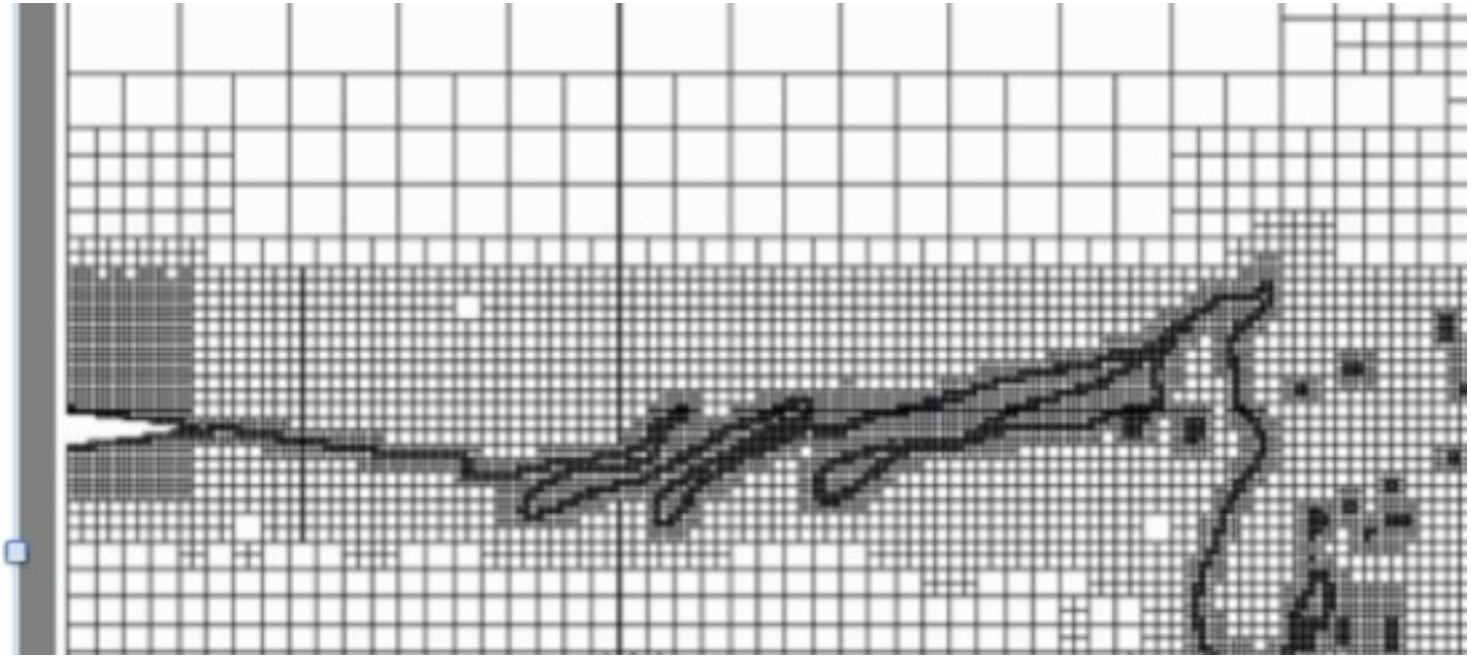
Surface tension

Treatment of surface tension by Continuous Surface Force (« CSF » method, Brackbill, Kothe and Zemach JCP 1993)

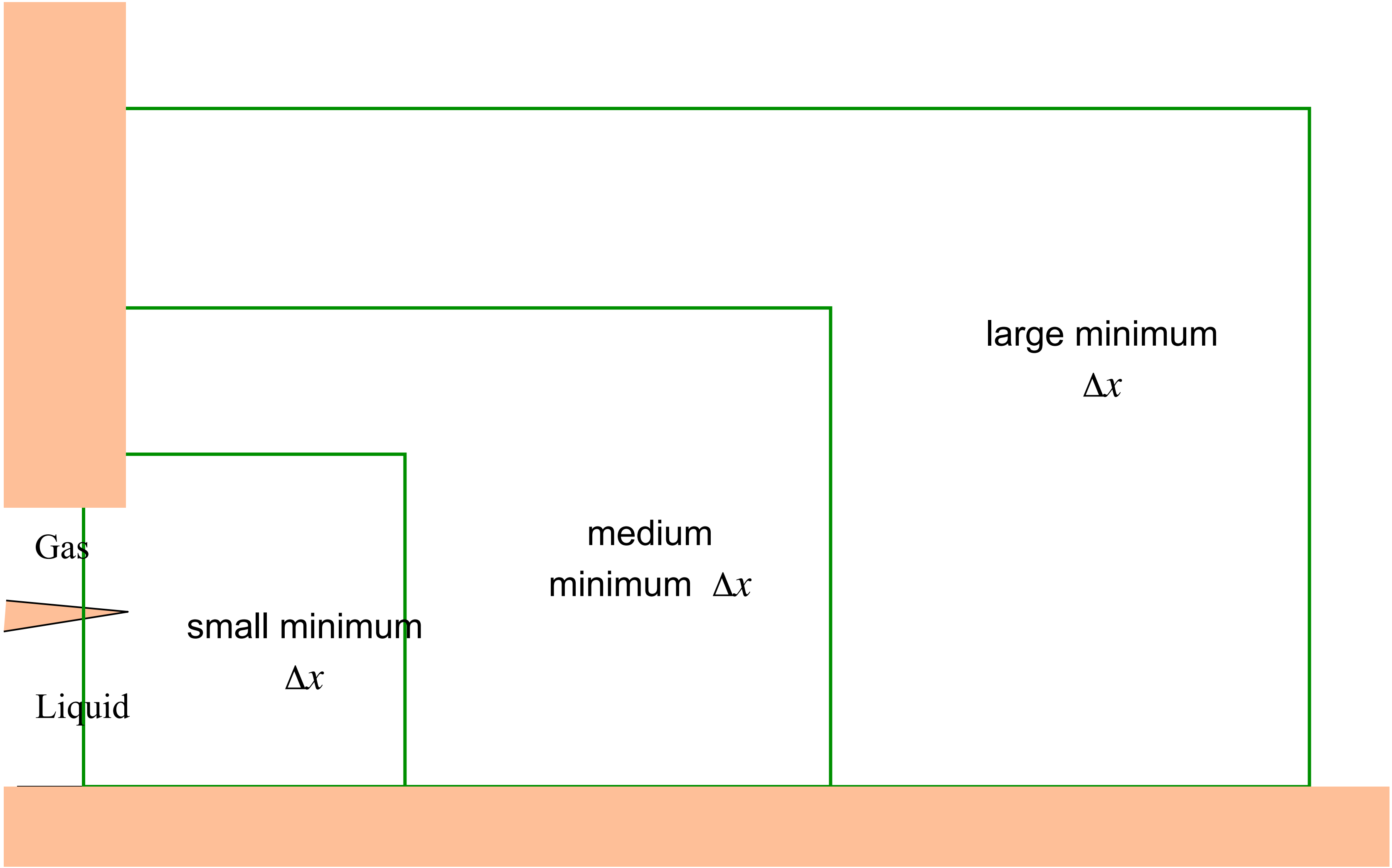
$$\sigma\kappa\,\mathbf{n}\delta_S \approx \sigma\kappa^h \nabla^h C$$

Many methods for κ .

Use Gerris flow solver (S. Popinet) with adaptive oct-tree and quad-tree grids

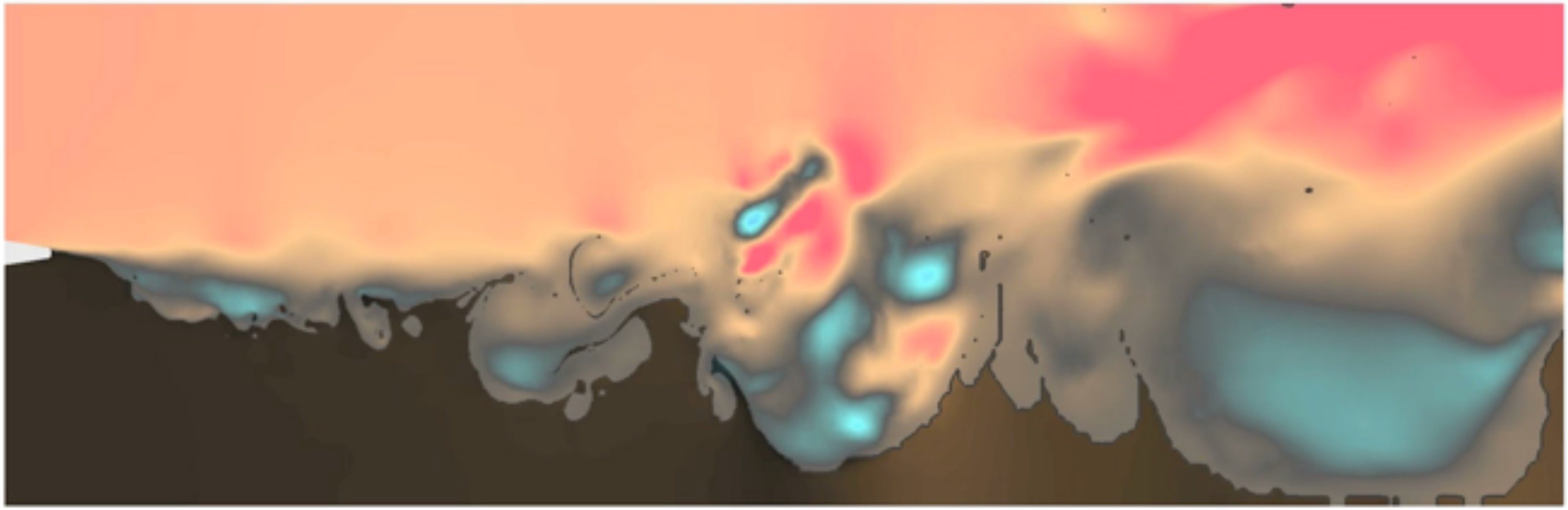


Navier-Stokes with variable minimum grid size according to a subdivision of the computational domain.



Simulation with a separator plate at density ratio (1/*r* = *100*)

m	r	Re_g	Re_l	We_g	We_l	M
0.017	0,01	2640	290	19	8	2,4



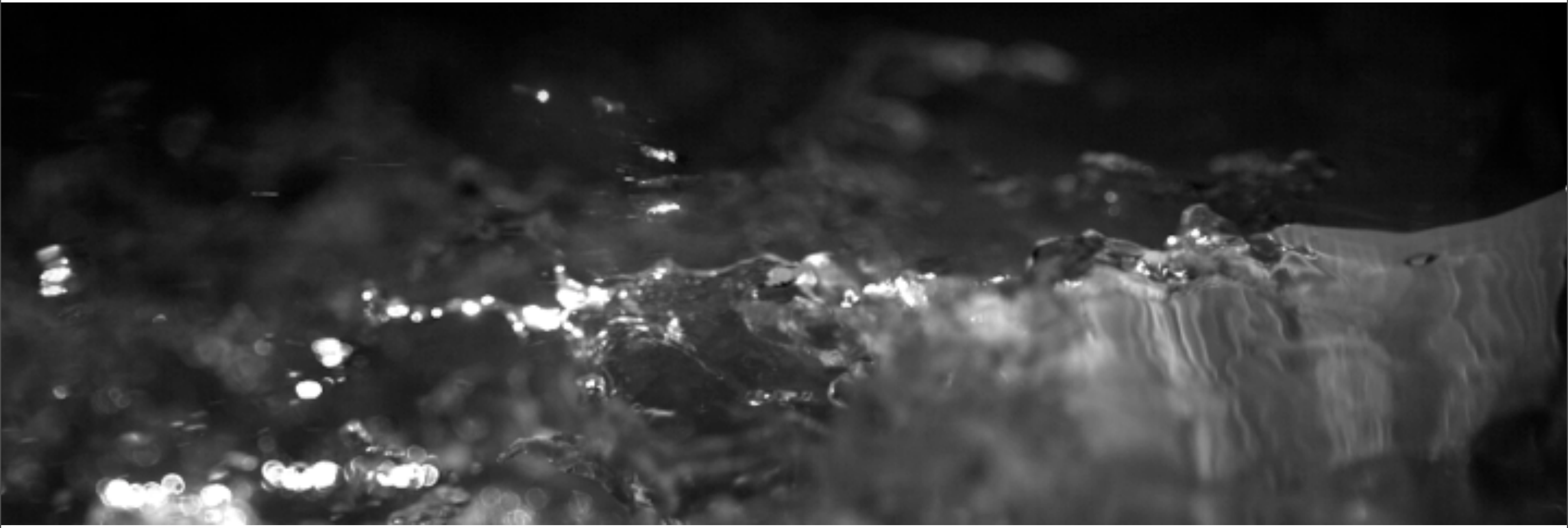
Movie by Daniel Fuster and Jérôme Hoepffner using the Gerris Flow solver

Simulation with a separator plate at density ratio ($1/r = 100$)

m	r	Re_g	Re_l	We_g	We_l	M
0.017	0,01	2640	290	19	8	2,4

Movie by Daniel Fuster and Jérôme Hoepffner using the Gerris Flow solver

Compare to experiments in Grenoble (Cartellier, Matas) . Flow from right to left.
Video with help of Jérôme Hoepffner and Jon Soundar.



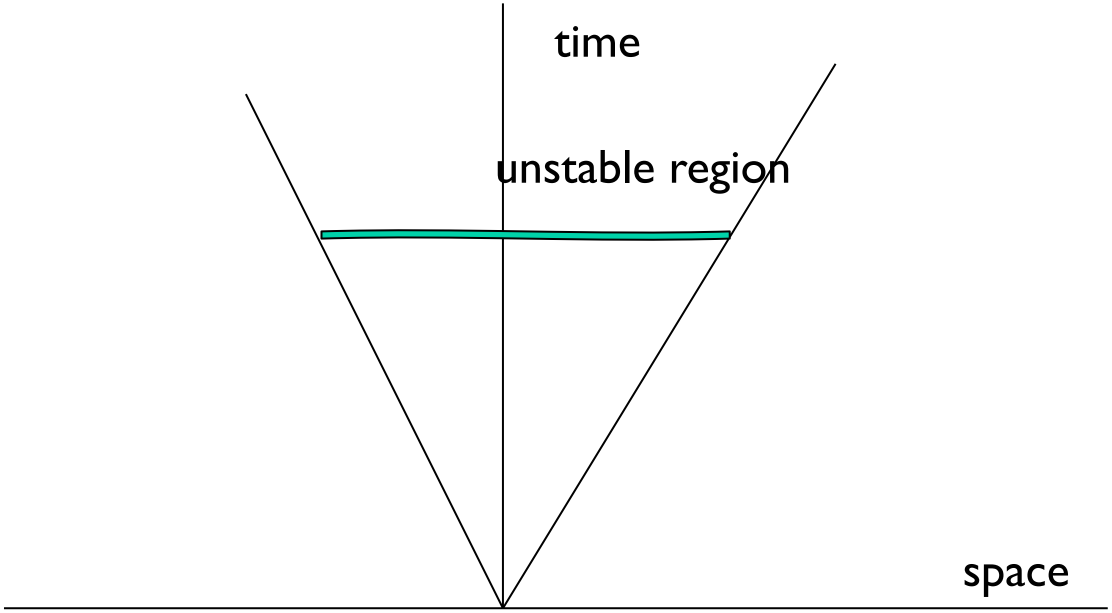
Compare to experiments in Grenoble (Cartellier, Matas) . Flow from right to left.
Video with help of Jérôme Hoepffner and Jon Soundar.

We need linear theory for spatially developping flows.

For that, we need to know what are absolute and convective instabilities !

Convective/absolute instabilities

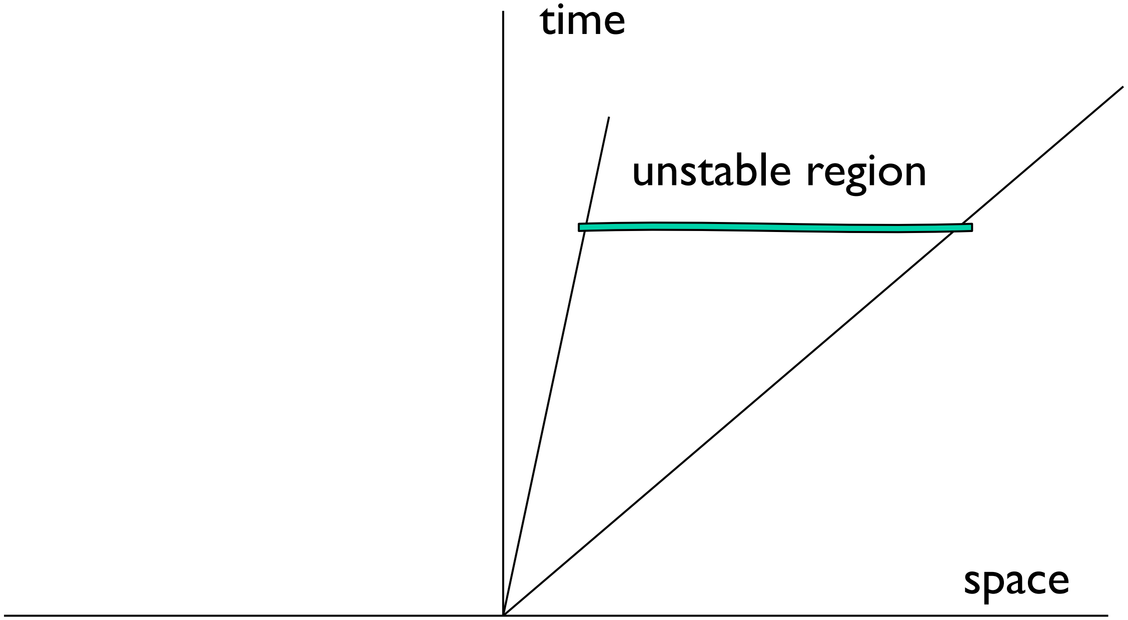
1) Absolute: a spatially localized perturbation at x=0 and t=0 grows in the entire space



corresponds to a well-defined oscillator frequency in the entire domain, a so-called « global mode »

Convective/absolute instabilities

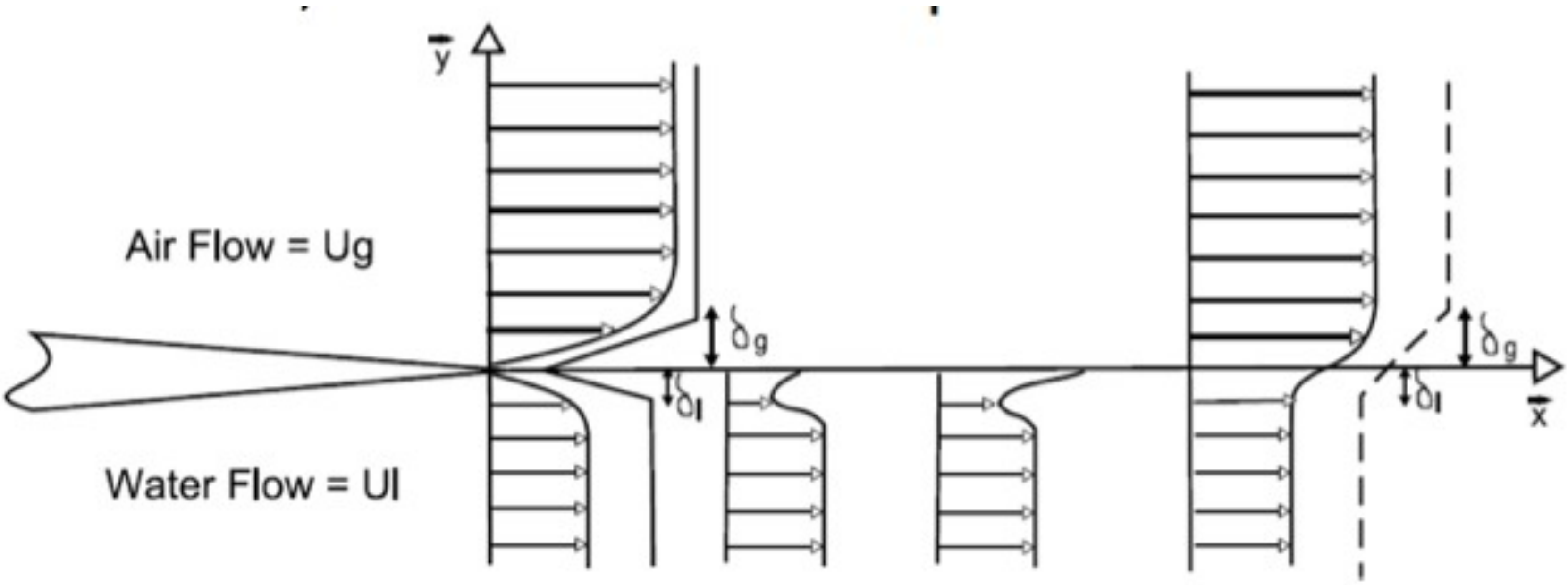
2) Convective instability: a spatially localized perturbation at t=0
is convected downstream with the flow



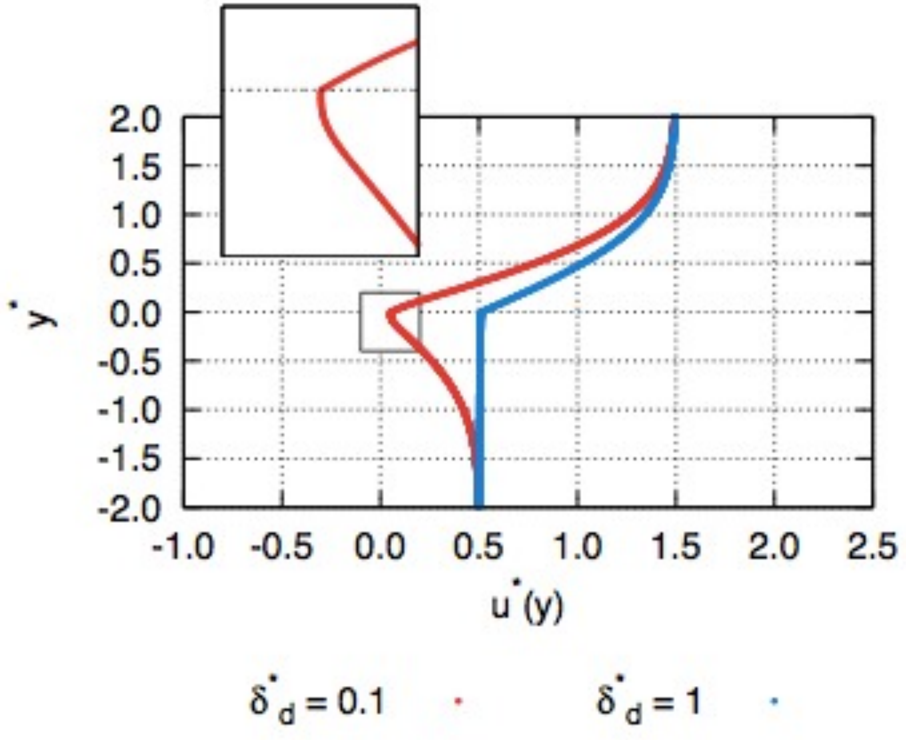
No single frequency is observed but instead, broadband noise is seen.
The system is seen to be a noise amplifier. Upstream turbulence matters

So what does linear theory say about our problem ? Is it convective or absolute ?

Linear theory has an enormous dependence on the wake flow correction.



Wake flow



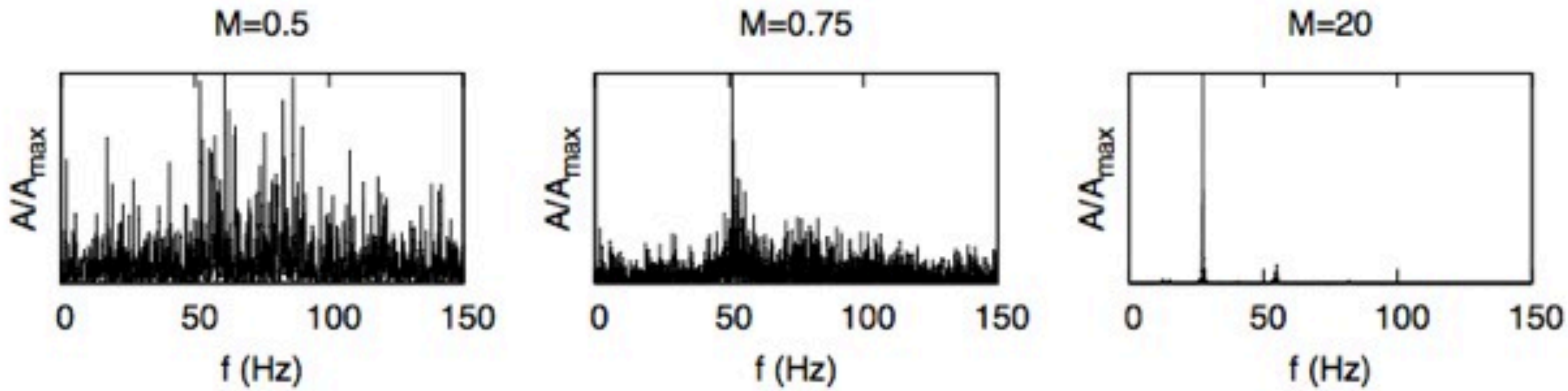
Simplified base flows

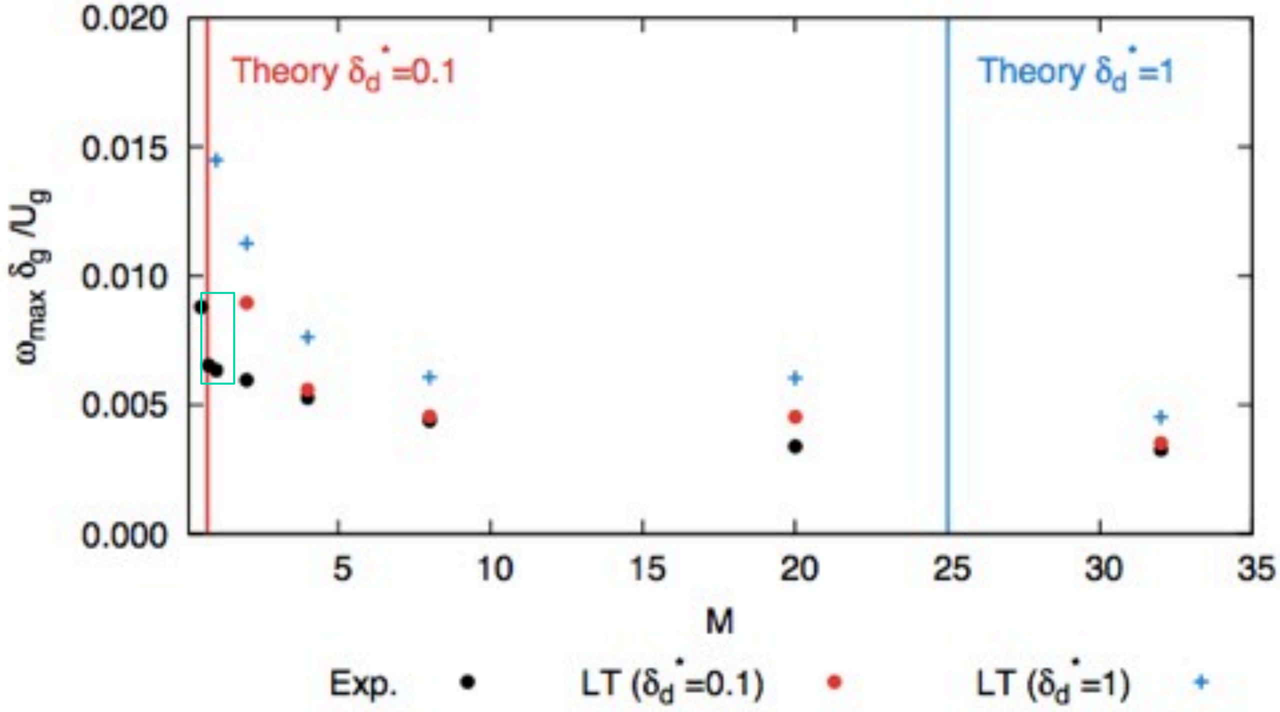
Most important parameter: momentum flux ratio
(or ratio of dynamic pressures)

$$M = \frac{\rho_g u_g^2}{\rho_l u_l^2}$$

Grenoble experiments: Cartellier, Matas, Marty

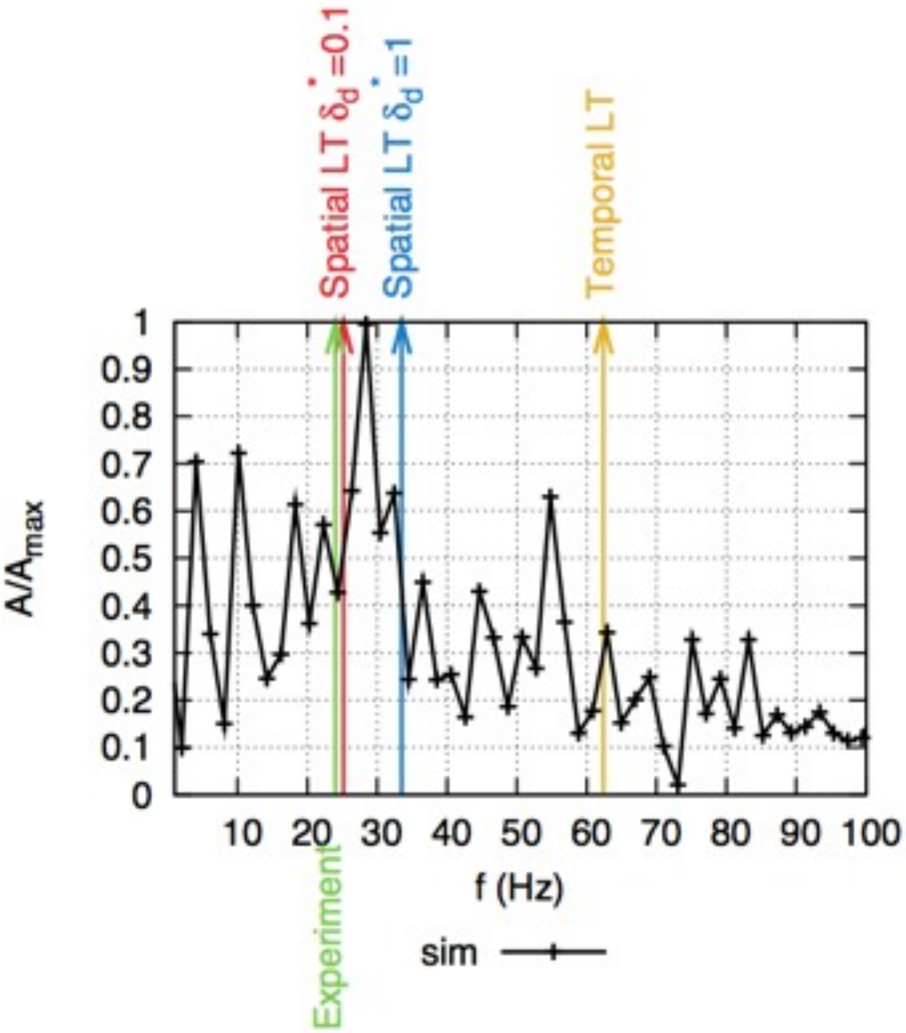
convective, noise amplifier ambiguous absolute, global mode



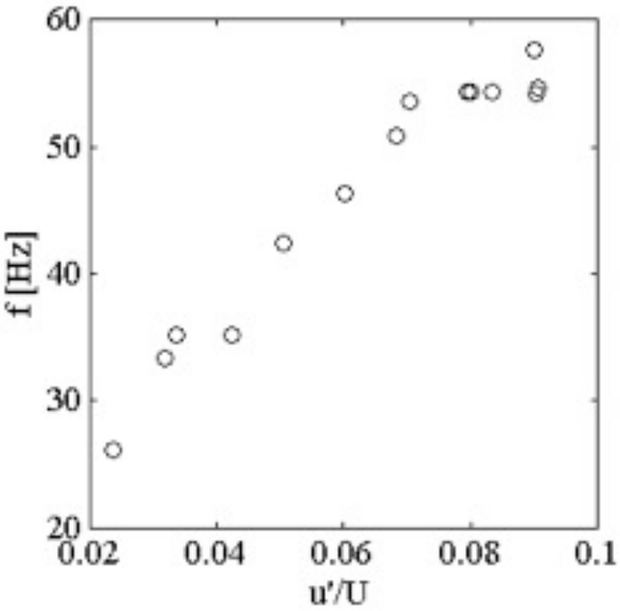


Now the ultimate test ! Compare :

- Experiments
- Numerics
- Linear theory


Spatial LT δ_d^*=0.1
Spatial LT δ_d^*=1
Temporal LT
Experiment
A/A$_{max}$
1
0.9
0.8
0.7
0.6
0.5
0.4
0.3
0.2
0.1
0
10
20
30
40
50
60
70
80
90
100
f (Hz)
sim

Primary instability of sheared interfaces is still a challenge however.



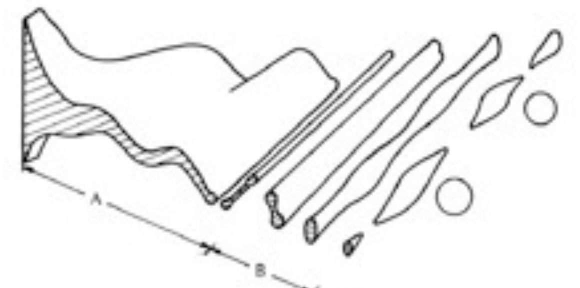
Strong influence of the turbulence level on the measured frequency.
Figure from Cartellier & Matas (LEGI)

2) 3D: How do 2D sheets break into 3D ligaments and droplets ?

How do 2D sheets break into 3D ligaments and droplets ?

Two universal mechanisms:

1) Cylinder (rim) + Rayleigh-Plateau instability



Dombrowski and Johns (1983)
Zhang, Li V., Brunet, P., Eggers, J. & Deegan, R. D. 2010

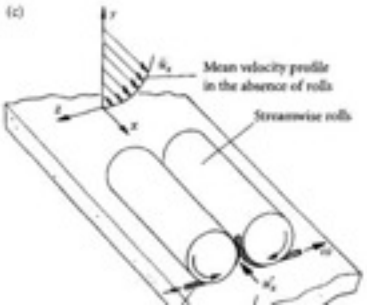
2) Hole formation

Roisman et al (2006)

Other more specialized mechanisms

for atomization

- streamline vortices pre-existing in the upstream bour



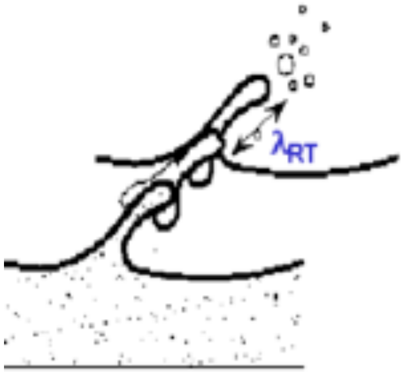
- Non-normal instability of two phase mixing layers (Yecko & Zaleski 2005) (Squire theorem does not apply in two-phase situation, the non-normal instability also leads to streamwise vortices)

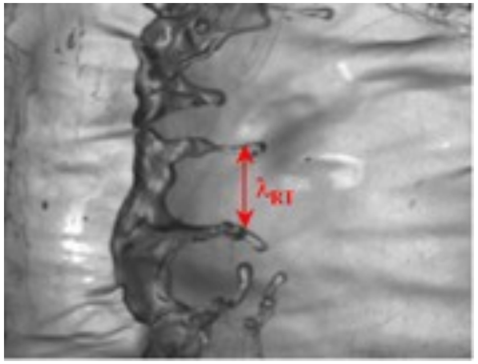
- for splashing

- Richtmyer-Meshkov instability (Gueyffier & Zaleski 1998)

What is observed in atomization ?

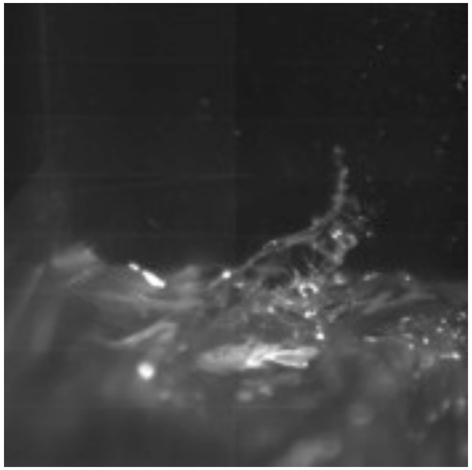
2D waves + attached ligaments formation



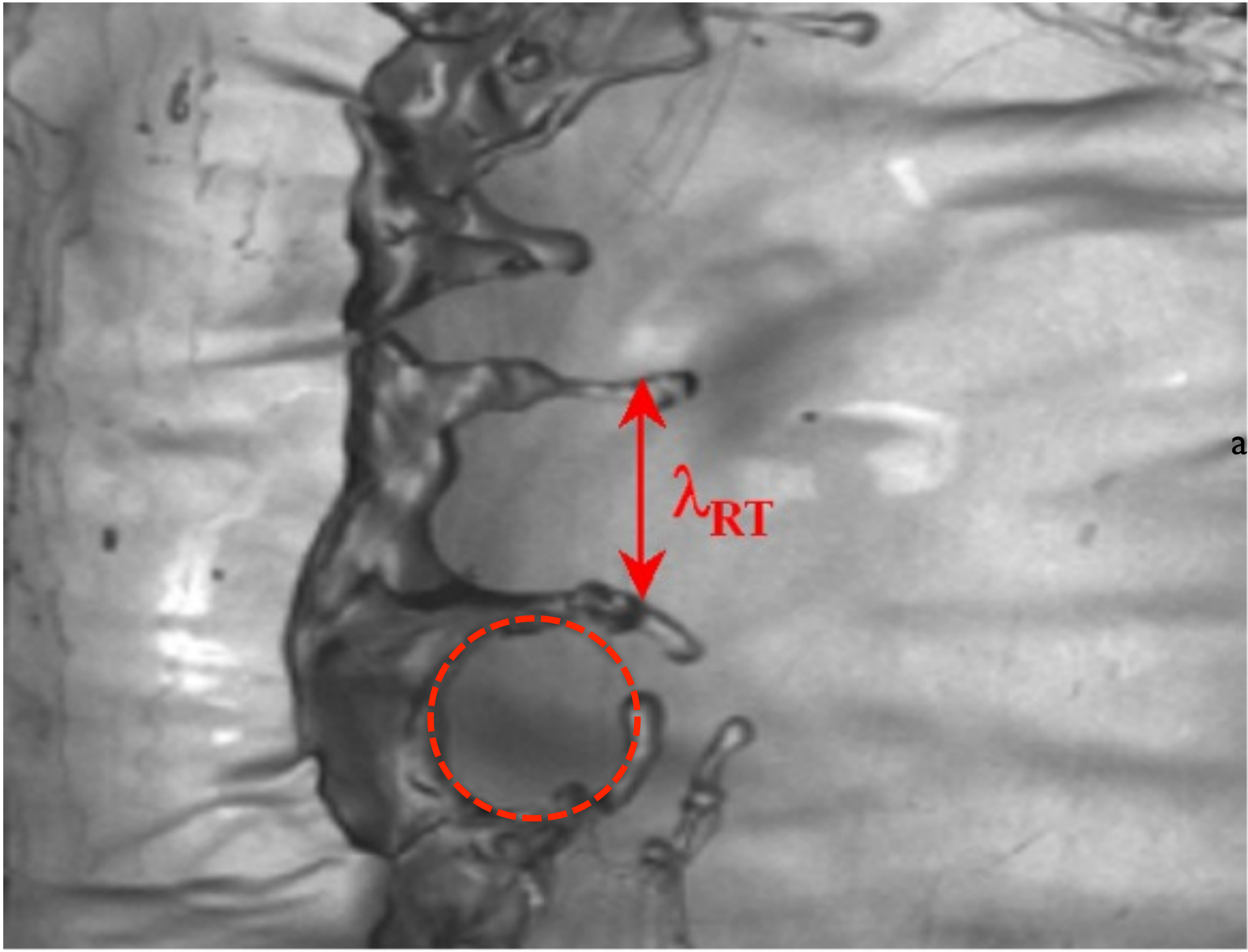


Photograph: Alain Cartellier and Jean-Philippe Matas

Holes + fishbone patterns

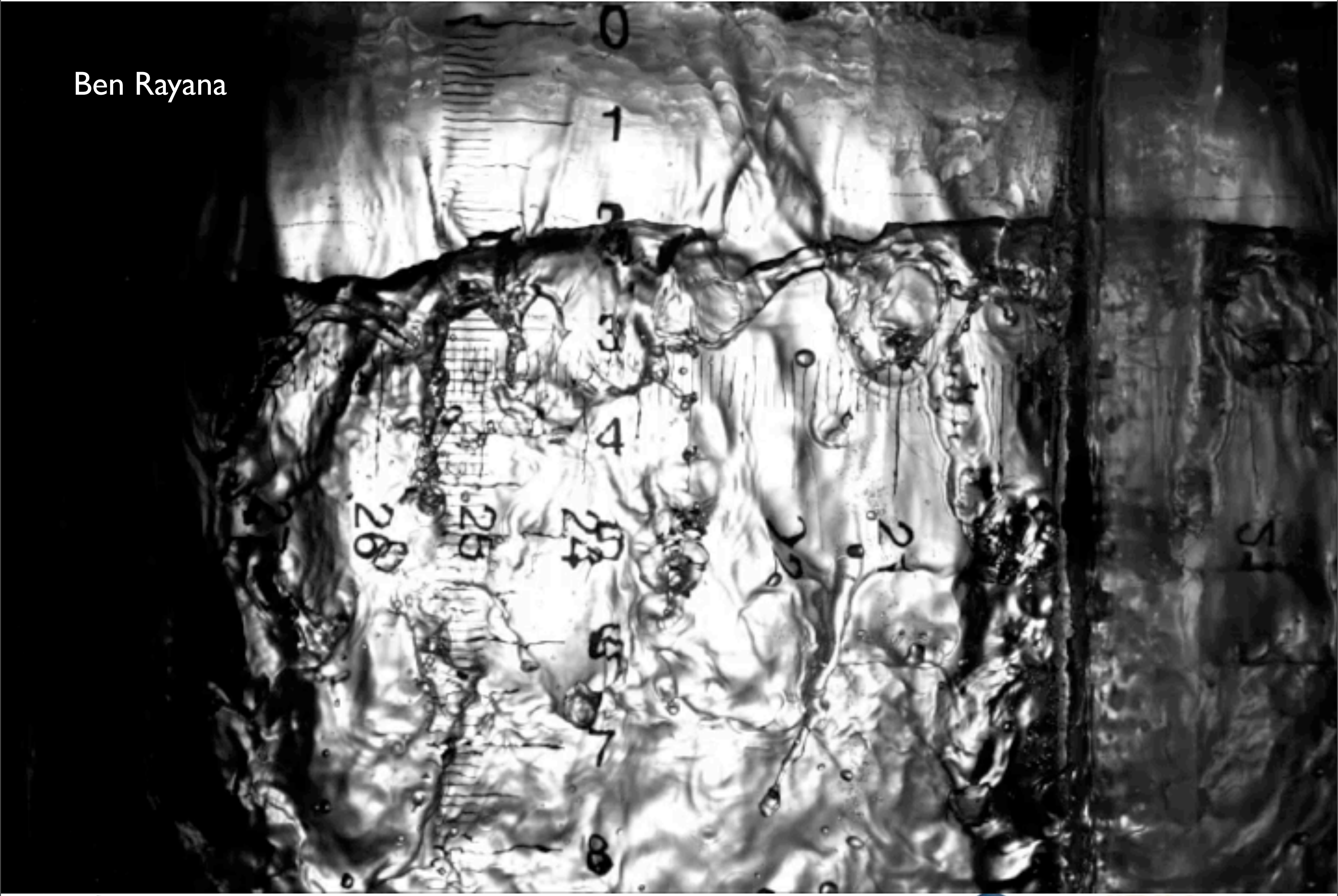


Photograph: Ludovic Raynal



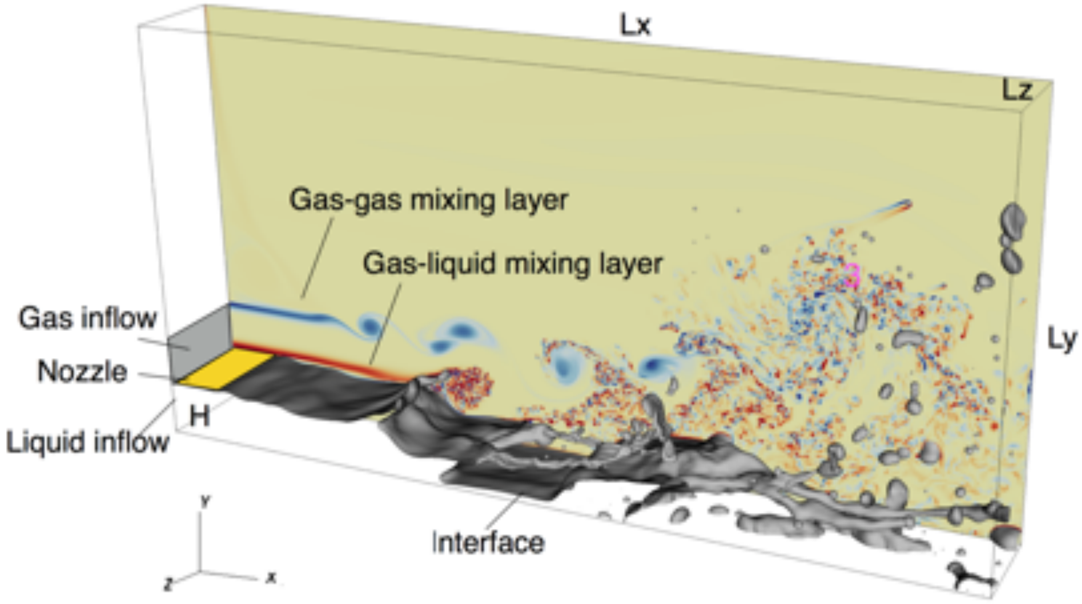
In fact
a) is also b)

Ben Rayana



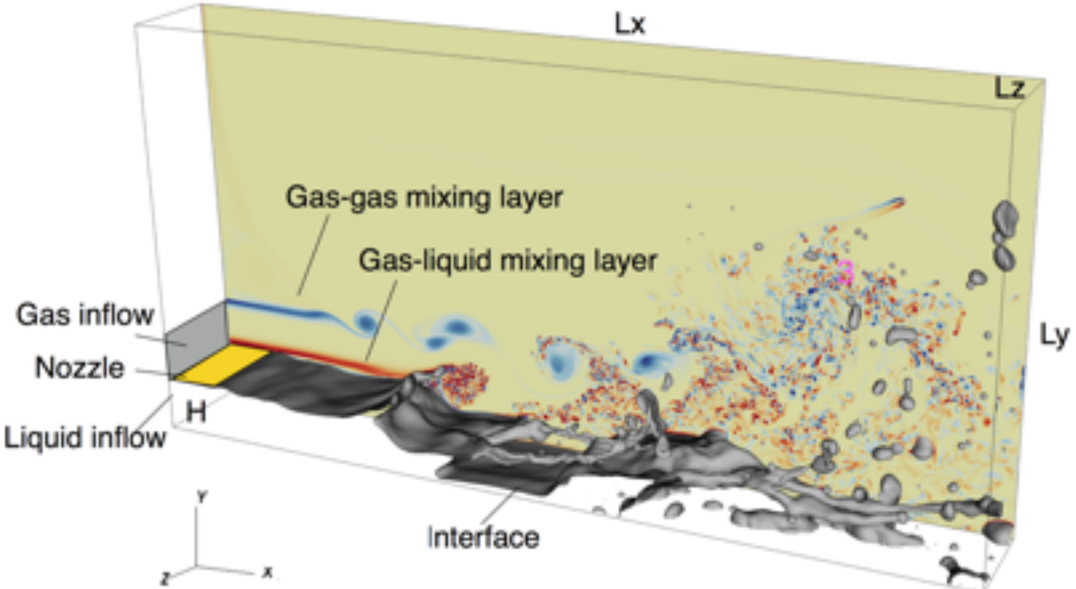
“A20” simplified case (Stanley Yue Ling) dimensional values

	Density (ρ) Kg/m^3	Viscosity (μ) Pa-s	Surface Tension (σ) N/m	Jet Height (H) mm	Boundary Layer (δ) mm	Injection Velocity (U) m/s
Gas	50	5E-05	0.05	0.8	0.1	10
Liquid	1000	1E-03		0.8	0.1	0.5



“A20” Benchmark: dimensionless values

M	$Re_{g,\delta}$	$Re_{g,H}$	$We_{g,\delta}$	r	m	v
$\frac{\rho_g U_g^2}{\rho_l U_l^2}$	$\frac{\rho_g U_g \delta}{\mu_g}$	$\frac{\rho_g U_g H}{\mu_g}$	$\frac{\rho_g U_g^2 \delta}{\sigma}$	$\frac{\rho_l}{\rho_g}$	$\frac{\mu_l}{\mu_g}$	$\frac{U_l}{U_g}$
20	1000	8000	10	20	20	20



- Turbulent gas flow
- Convective instability *Fuster et al. 2013, Otto et al. 2013*
- "Strong" atomization

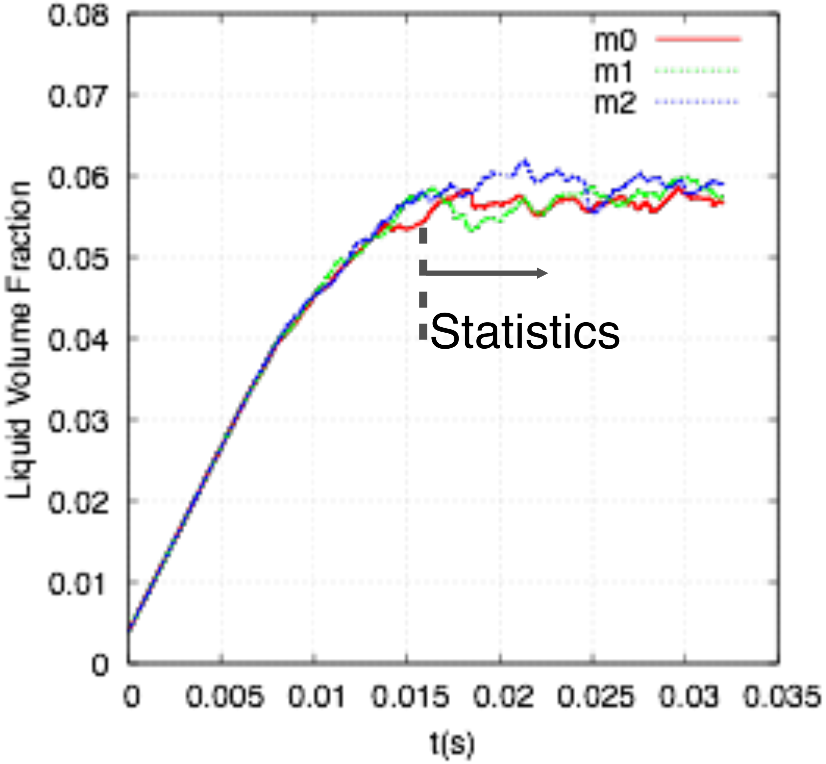
Simulation Cases

Domain: Lx=16H, Ly=8H, Lz=2H; End-Time: $t/(H/U_g)=400$

Cases	h(μm)	H/h	# of cells	# of time steps	Total CPU time (hr)
M0	25	32	8.4 Million	4.9E+04	2.5E+03
M1	12.5	64	67 Million	1.0E+05	4.3E+04
M2	**6.25**	**128**	**537 Million**	**2.2E+05**	**5.0E+05**
M3	3125	256	4 Billion	4.5E+05	8.0E+06

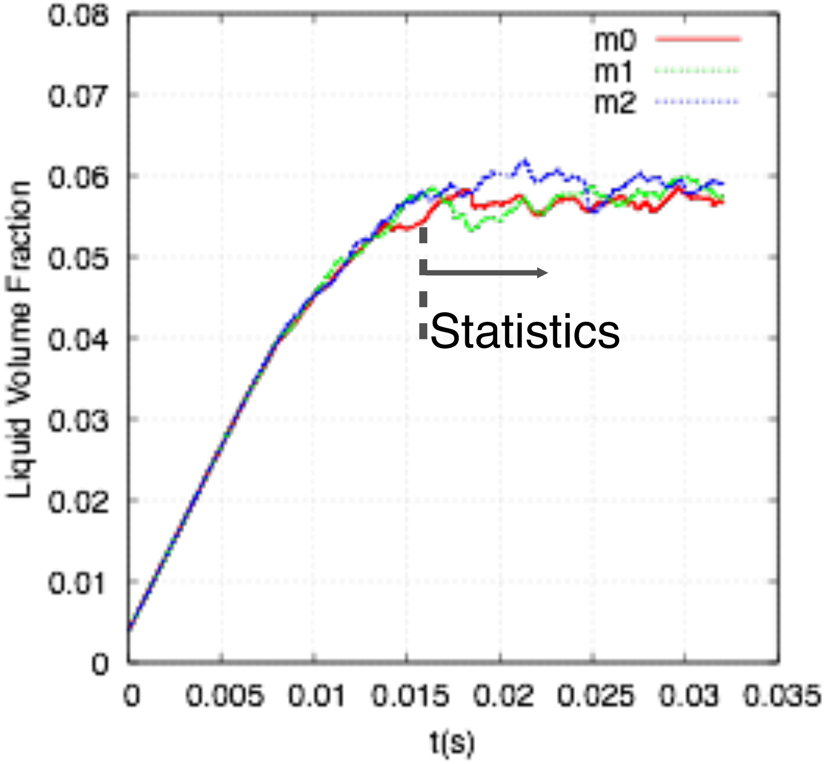
CPU time estimate based on performance on TGCC-CURIE machine

Liquid Volume Fraction

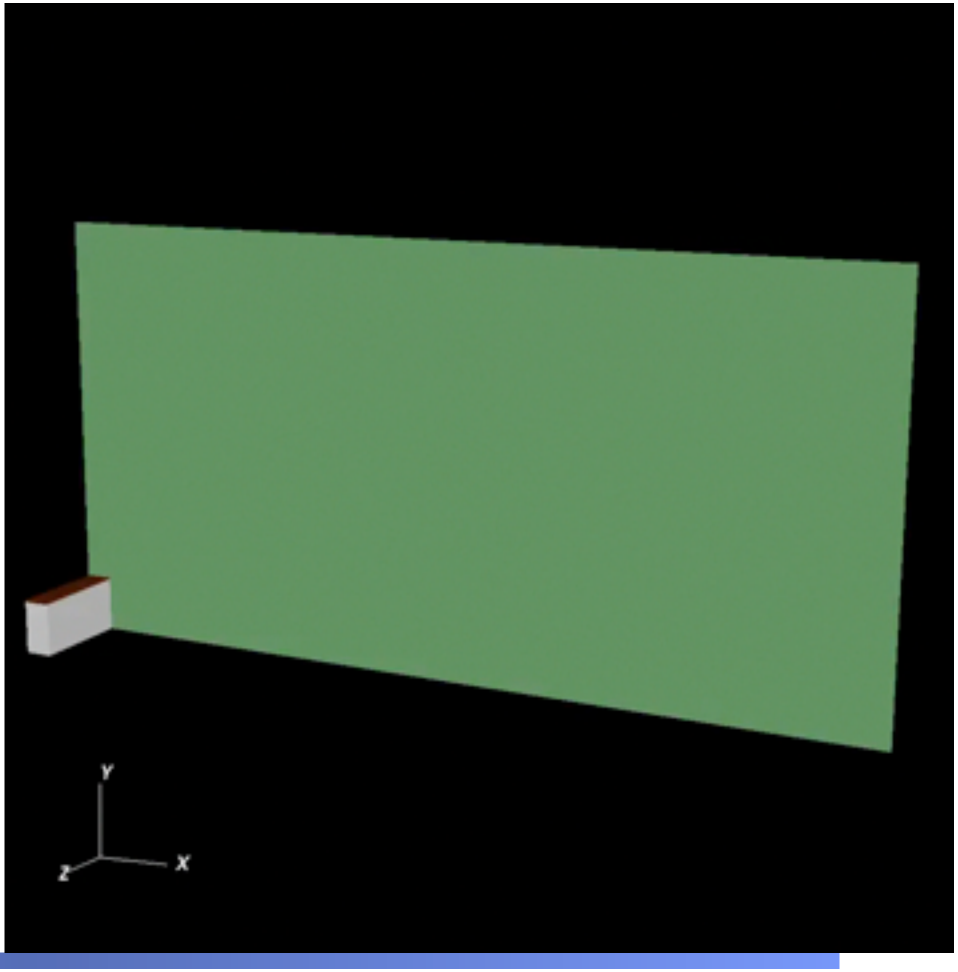


Statistics

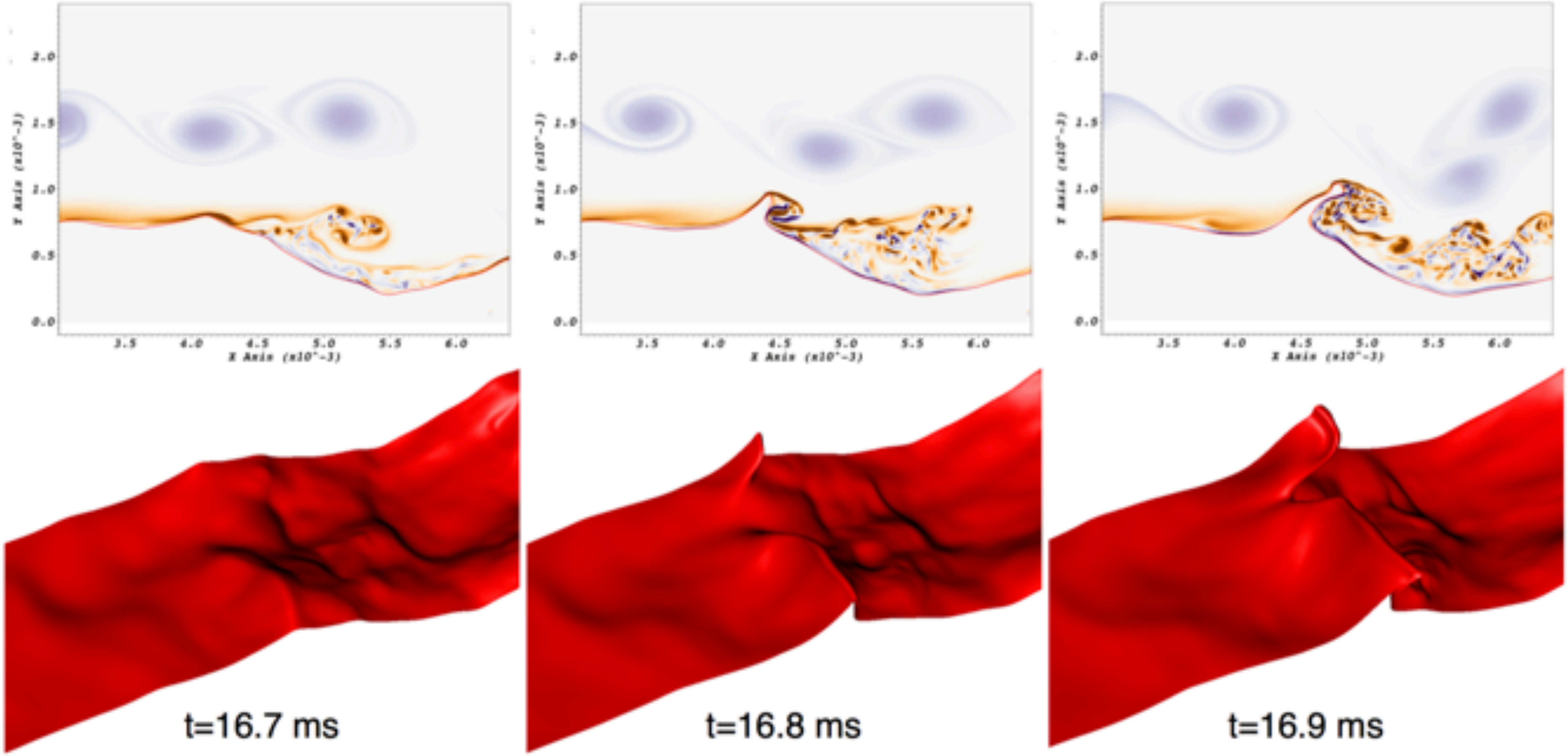
Liquid Volume Fraction



Statistics



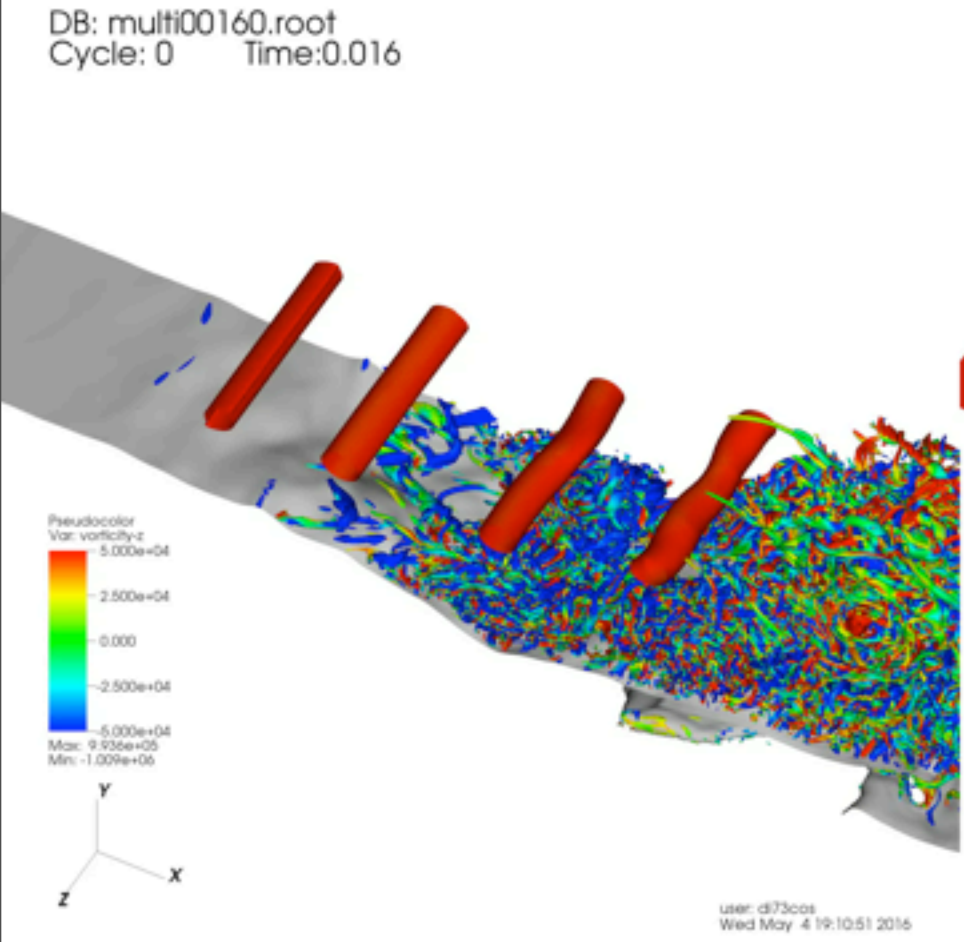
Sheet and rim formation dynamics

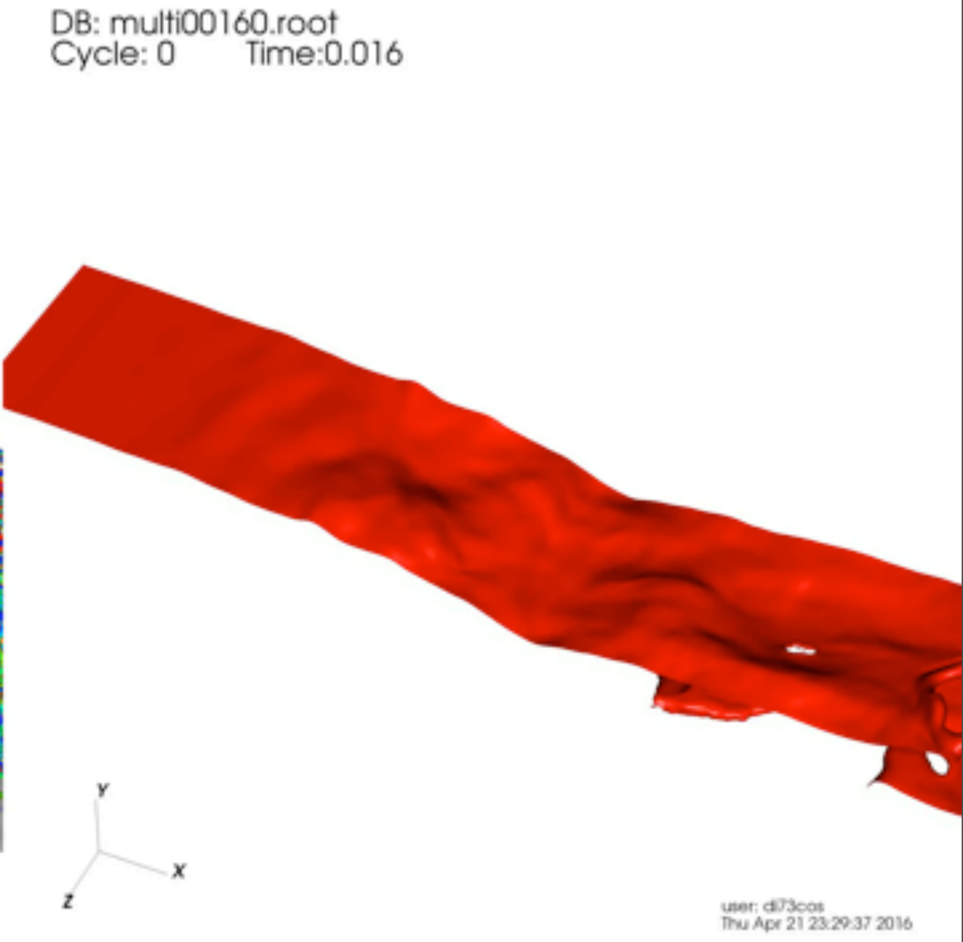


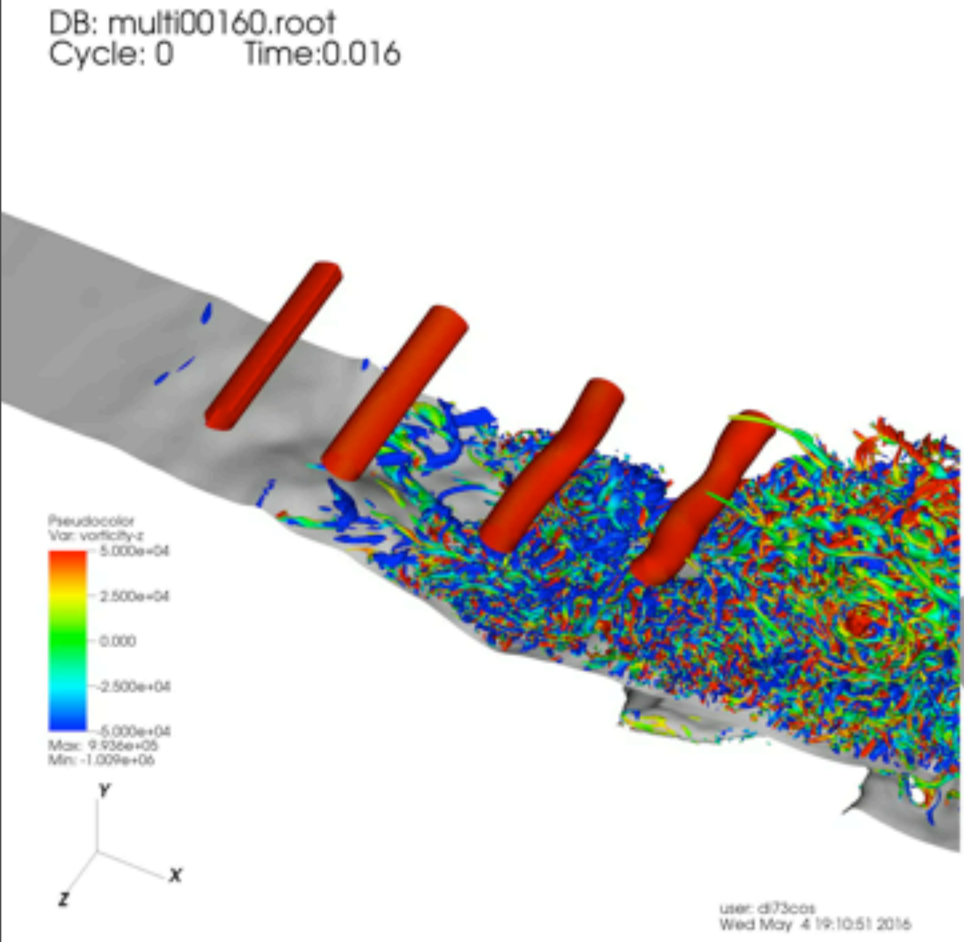
t=16.7 ms

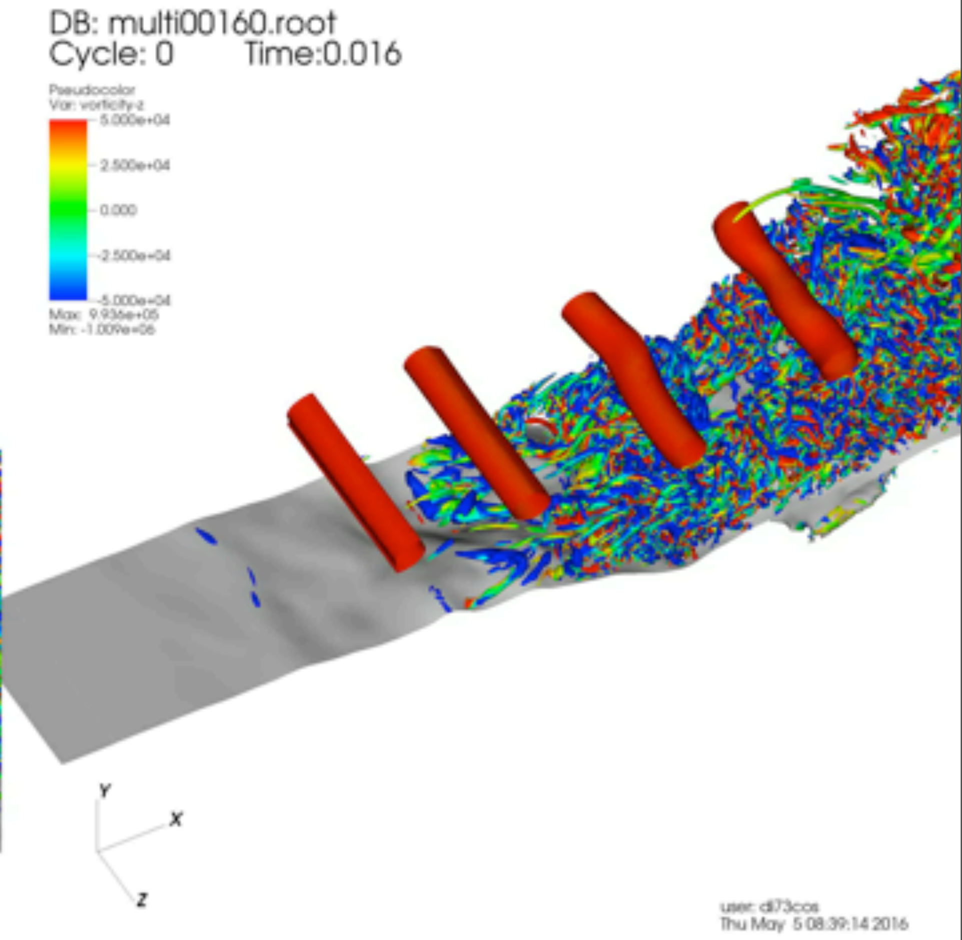
t=16.8 ms

t=16.9 ms

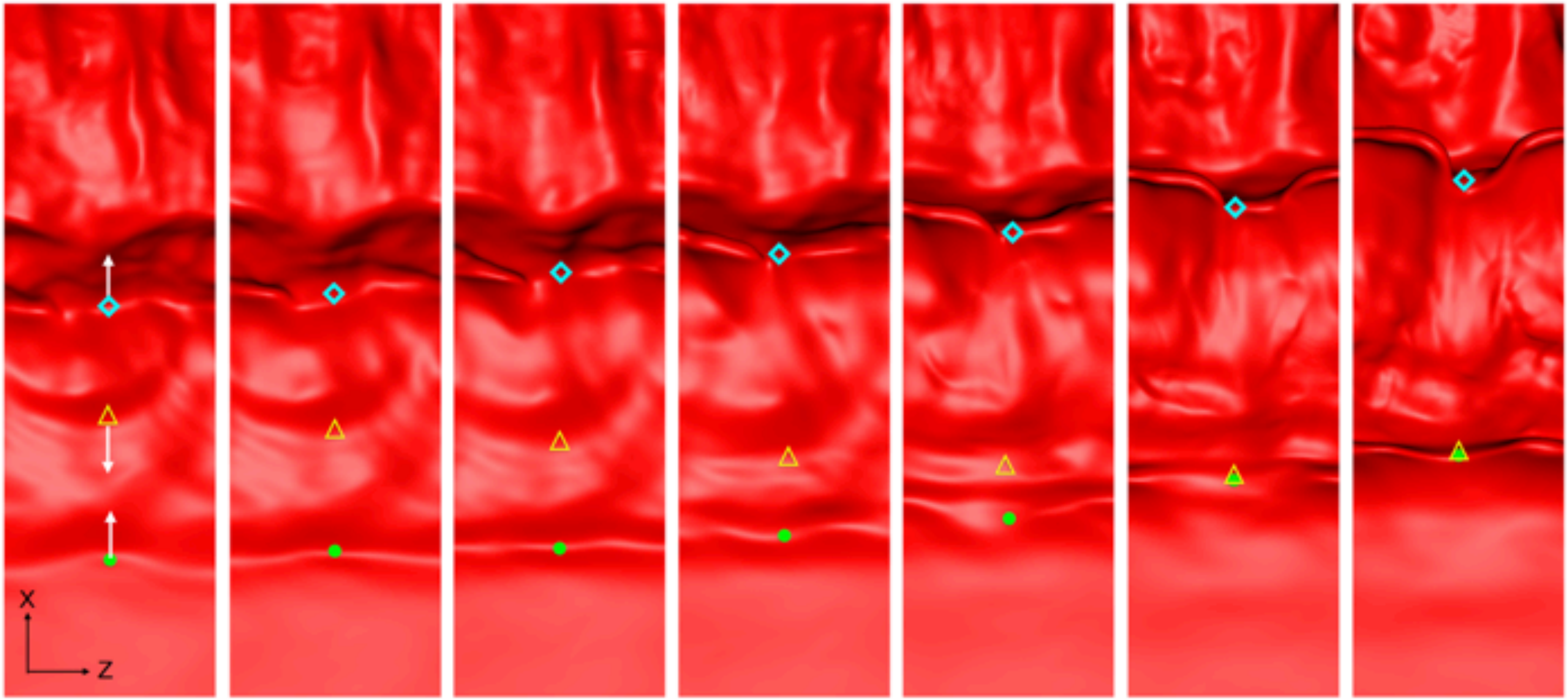

DB: multi00160.root
Cycle: 0 Time:0.016
Pseudocolor
Var: vorticity-z
5.000e+04
2.500e+04
0.000
-2.500e+04
-5.000e+04
Max: 9.936e+05
Min: -1.009e+06
Y
X
Z
user: di73cos
Wed May 4 19:10:51 2016

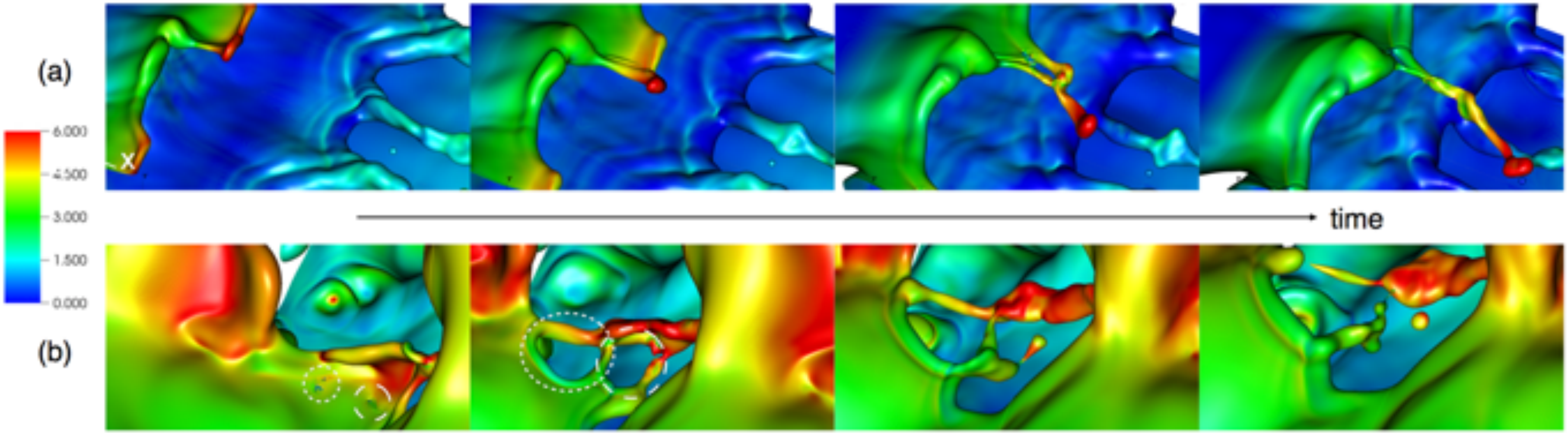

DB: multi00160.root
Cycle: 0 Time:0.016
Y
X
Z
user: di73cos
Thu Apr 21 23:29:37 2016


DB: multi00160.root
Cycle: 0 Time:0.016
Pseudocolor
Var: vorticity-z
5.000e+04
2.500e+04
0.000
-2.500e+04
-5.000e+04
Max: 9.936e+05
Min: -1.009e+06
Y
X
Z
user: di73cos
Wed May 4 19:10:51 2016

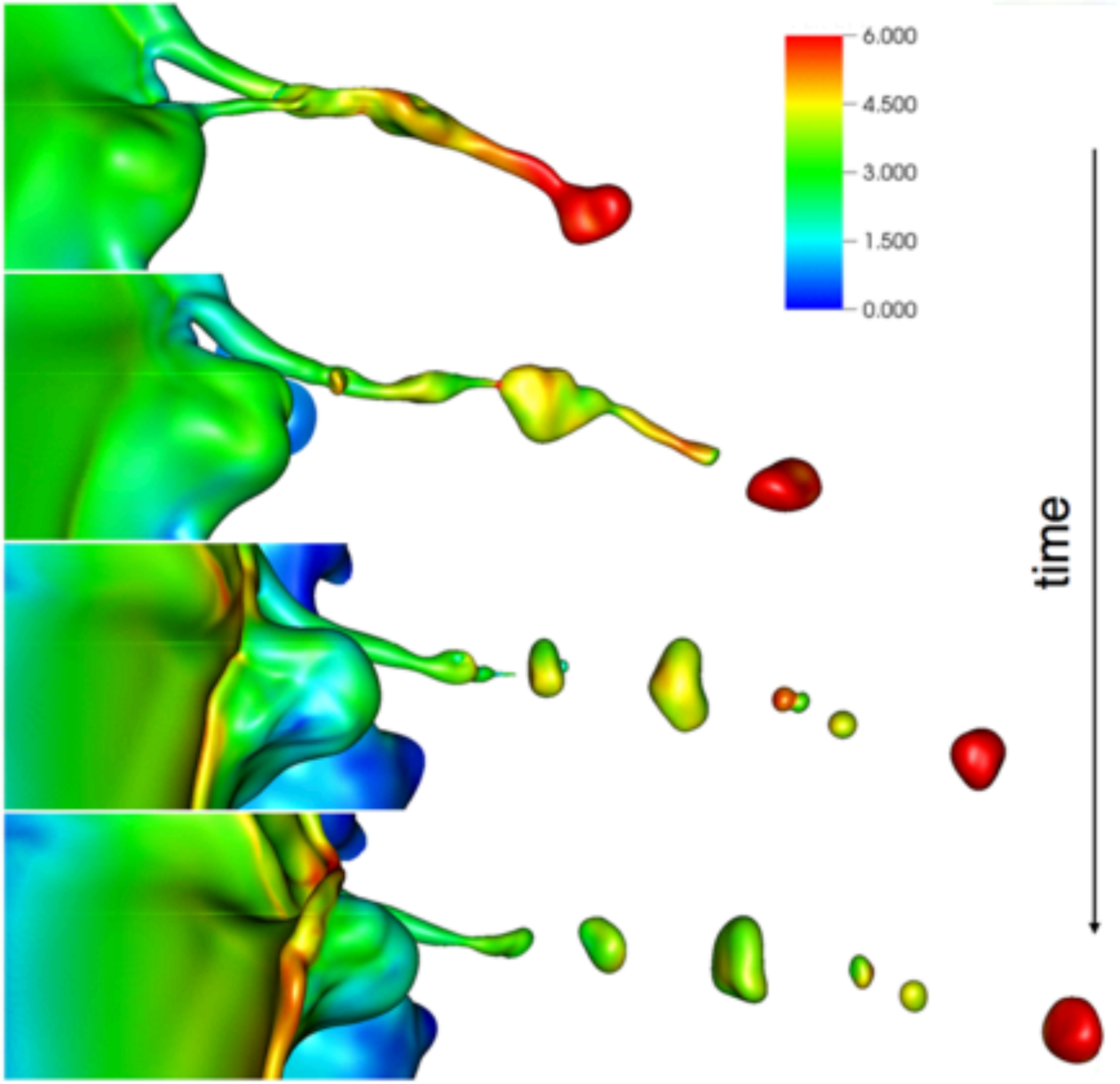

DB: multi00160.root
Cycle: 0 Time:0.016
Pseudocolor
Var: vorticity-z
5.000e+04
2.500e+04
0.000
-2.500e+04
-5.000e+04
Max: 9.936e+05
Min: -1.009e+06
Y
X
Z
user: di73cos
Thu May 5 08:39:14 2016

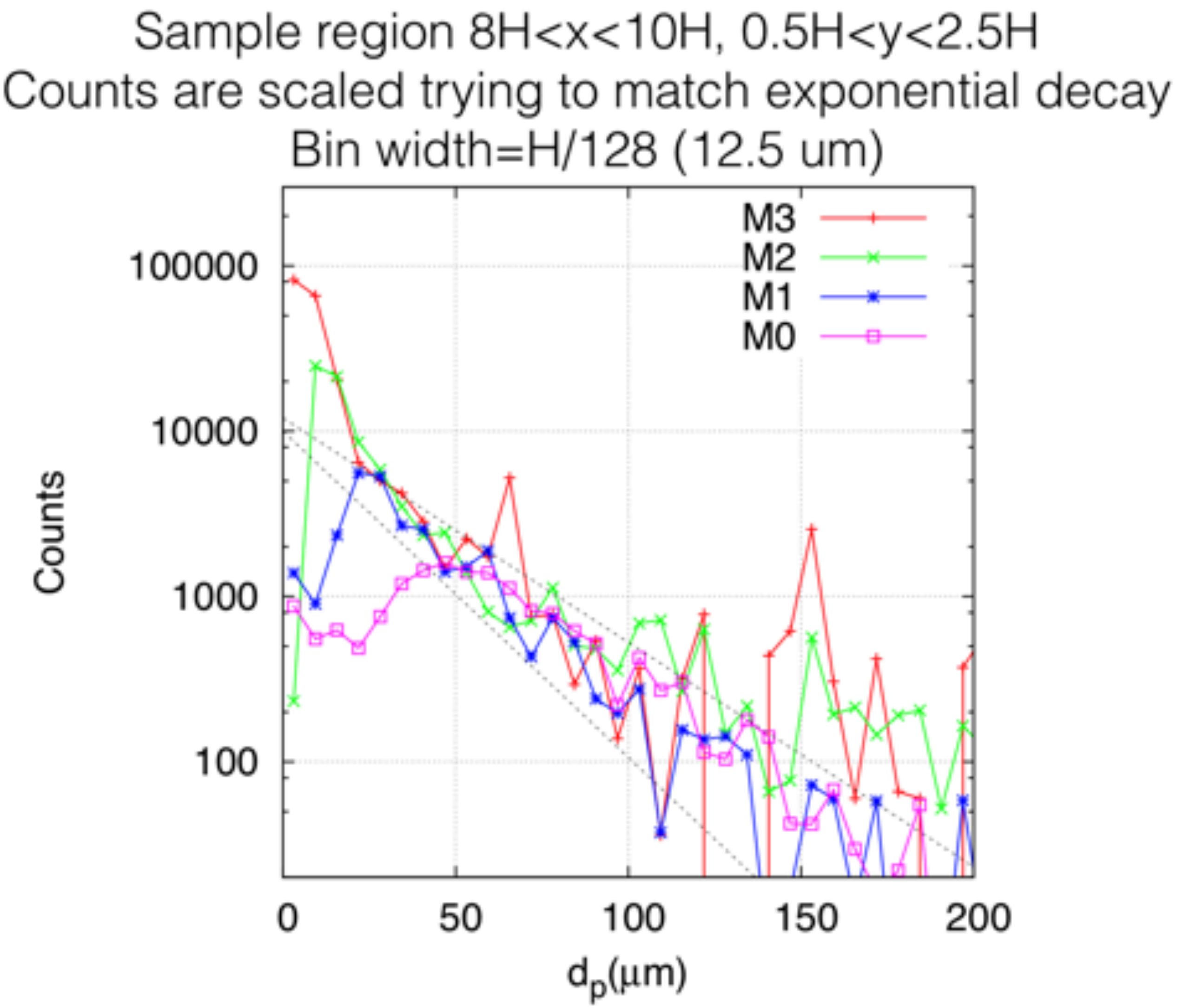
Interfacial wave interaction





Ligaments formation due to (a) fingering from the tip of liquid sheet and (b) hole formation in the liquid sheet. The color on the interface indicates the streamwise velocity.


6.000
4.500
3.000
1.500
0.000
time



Sample region 8H<x<10H, 0.5H<y<2.5H
Counts are scaled trying to match exponential decay
Bin width=H/128 (12.5 um)

1) Is it possible to do a real Direct Numerical Simulation of atomisation, resolving all the scales ?

Not yet

2) What can we learn from these very detailed simulations ?

How to look at the experiment again

(mechanisms much more complex than expected).

Acknowledgement

AGENCE NATIONALE DE LA RECHERCHE
ANR

GENCI
GRAND EQUIPEMENT NATIONAL DE CALCUL INTENSIF

PRACE

Y. Ling, D. Fuster, G
Tryggvason, S. zalesk
"Spray formation: ar
inverse cascade",
arXiv:1511.04234

Thank you!

Acknowledgement

AGENCE NATIONALE DE LA RECHERCHE ANR

GENCI

PRACE

Y. Ling, D. Fuster, G
Tryggvason, S. zalesk
"Spray formation: ar
inverse cascade",
arXiv:1511.04234

Backup Slides

This simulation is

- slow
- unstable
- does not parallelize properly
- does not have Lagrangian particles

Challenge: do better.

Two approaches:

1) a much more efficient approach to computing on octree: Basilisk by S. Popinet: http://basilisk.fr

2) or a very simple code on regular grids: Parissimulator by
Scardovelli, Ling, Tryggvason, Zaleski
http://parissimulator.sf.net

Speed issues

Add Lagrangian Point Particles (LPP) that obey a point-particle equation

$$\frac{\mathrm{d}\mathbf{u}_p}{\mathrm{d}t} = \mathbf{F}_p\left[\mathbf{u}_p, \mathbf{u}_f(\mathbf{x})\right]$$

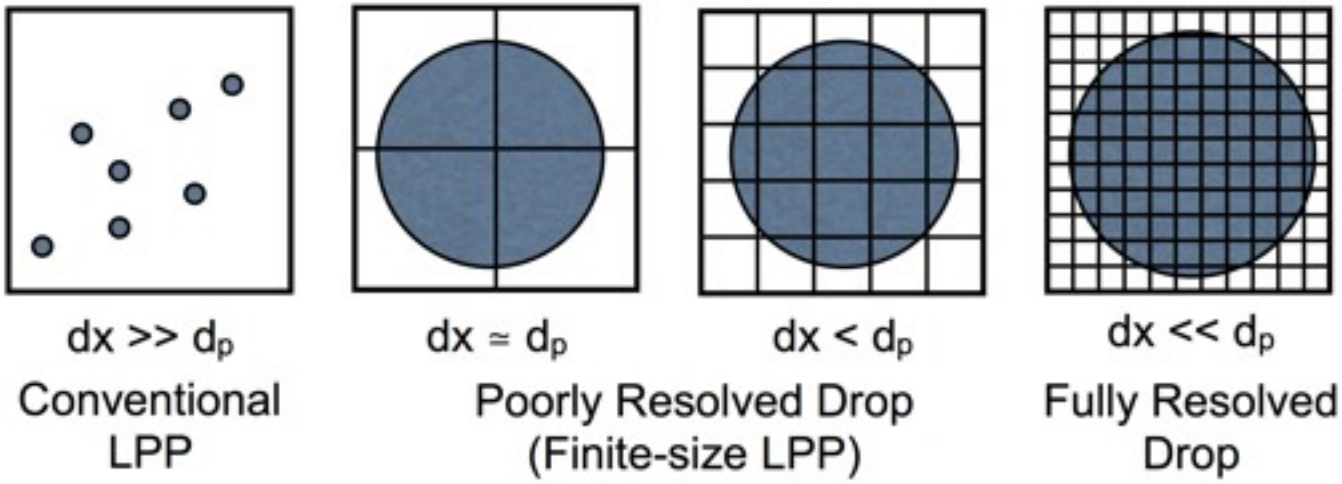
The force is determined by the surrounding carrier fluid velocity field $\mathbf{u}_f(\mathbf{x})$
It reacts on the Navier-Stokes equation through a smoothing Kernel G :

$$\rho\, \mathrm{D}\mathbf{u} / \mathrm{Dt} = -\nabla p + \nabla \cdot (2\mu \mathbf{D}) + \sigma\kappa\delta_S \mathbf{n} - \mathbf{F}_p * \mathbf{G}$$

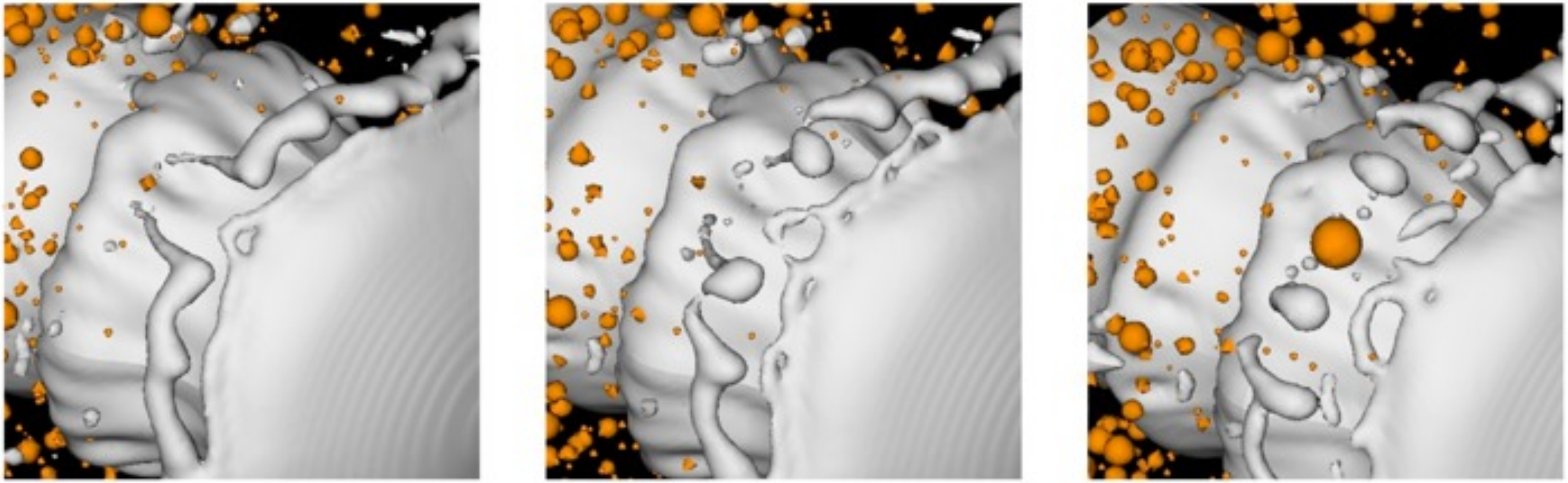
where the strain-rate tensor **D** is

$$D_{ij} = \frac{1}{2}\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right).$$

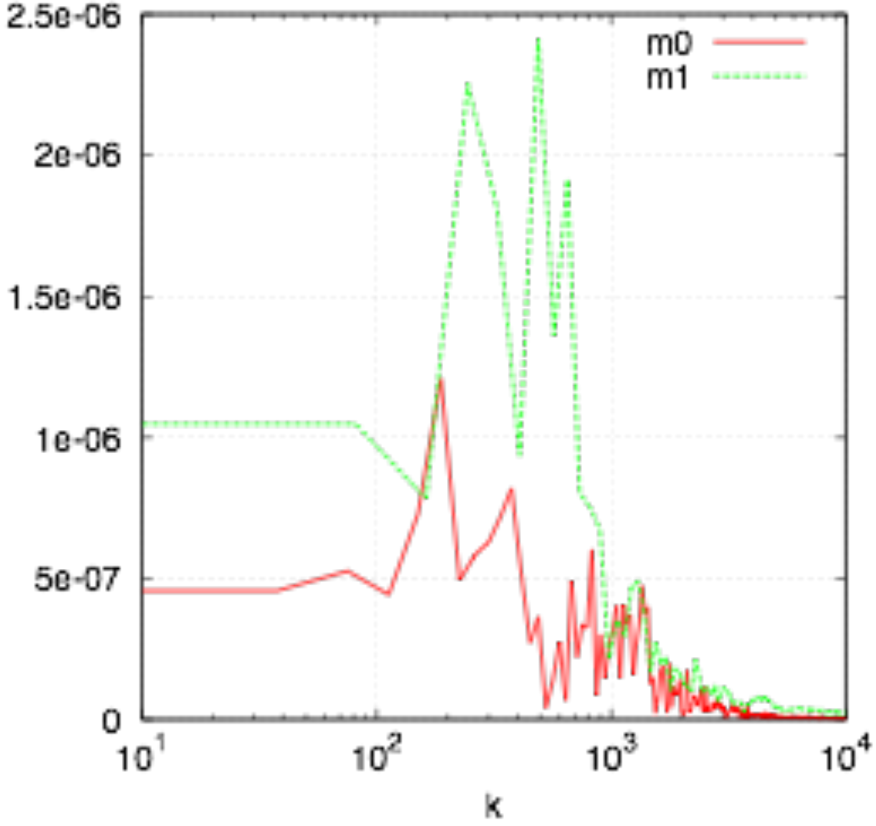
The choice of resolved interface or LPP modelling depends on the type of simulation and on grid resolution



VOF to LPP conversion - High Reynolds – CORIA (Berlemont) jet

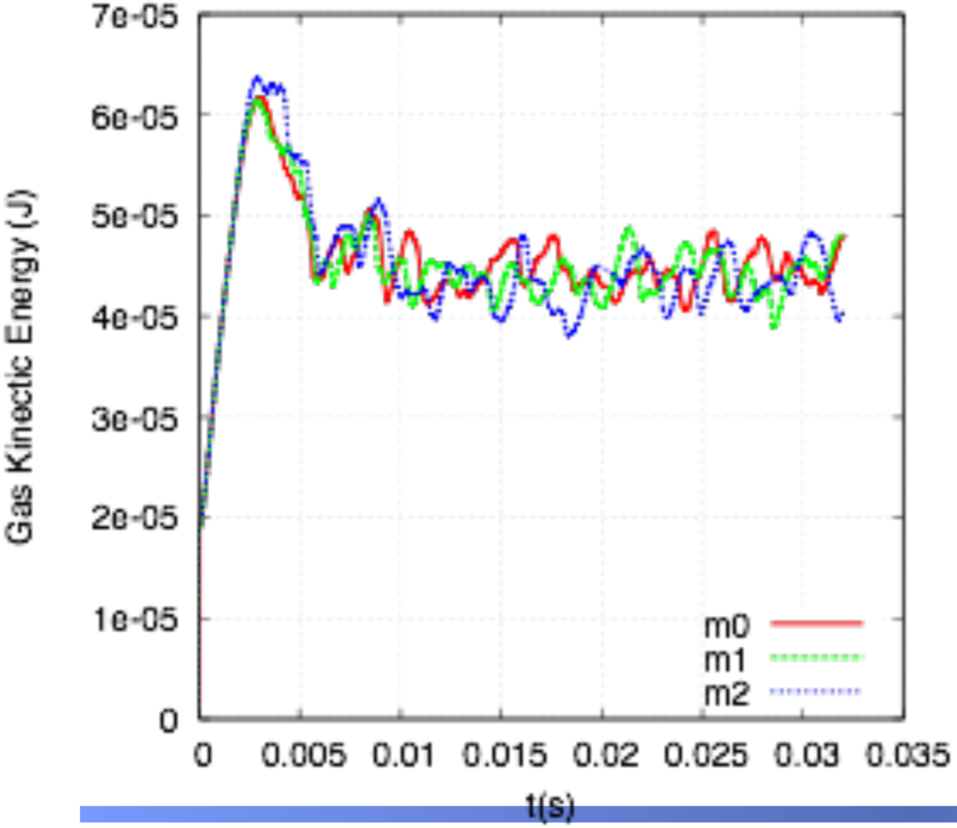


time

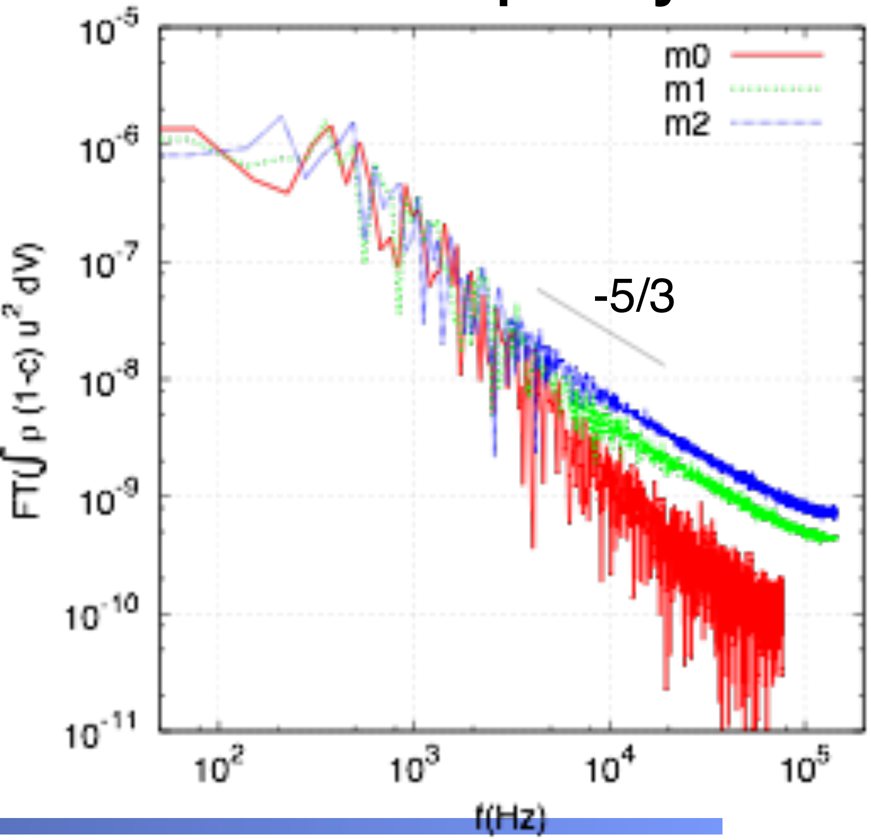

2.5e-06
2e-06
1.5e-06
1e-06
5e-07
0
10^1
10^2
10^3
10^4
k
m0
m1

Total Gas Kinetic Energy

Time

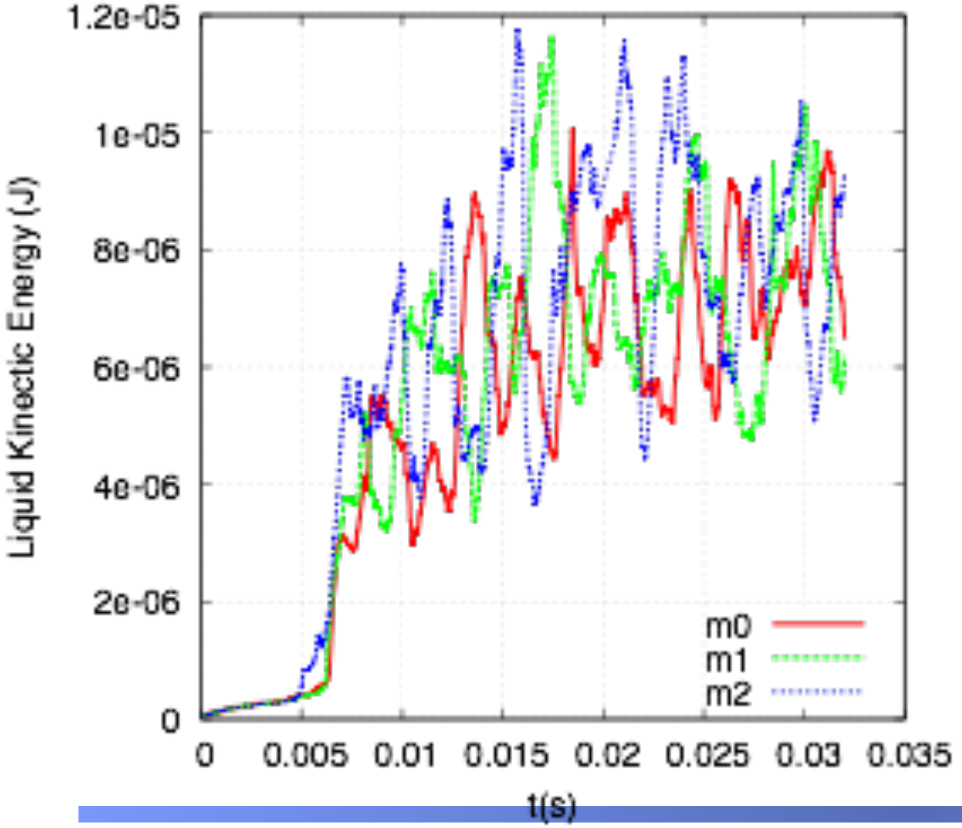


Frequncy

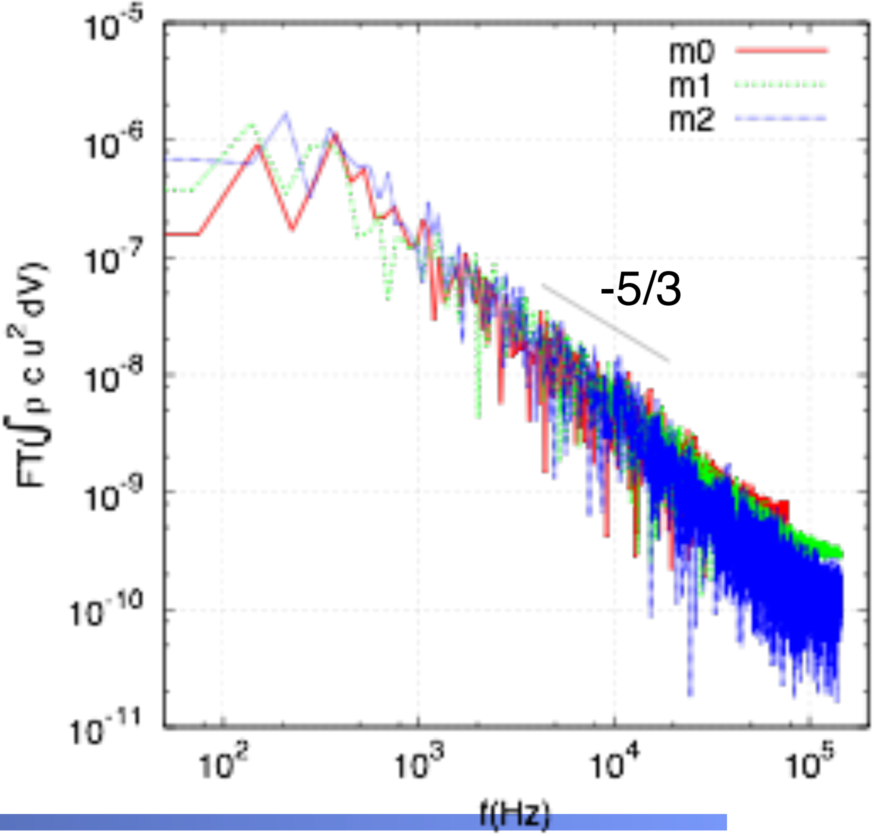


Total Liquid Kinetic Energy

Gas Kinetic Energy

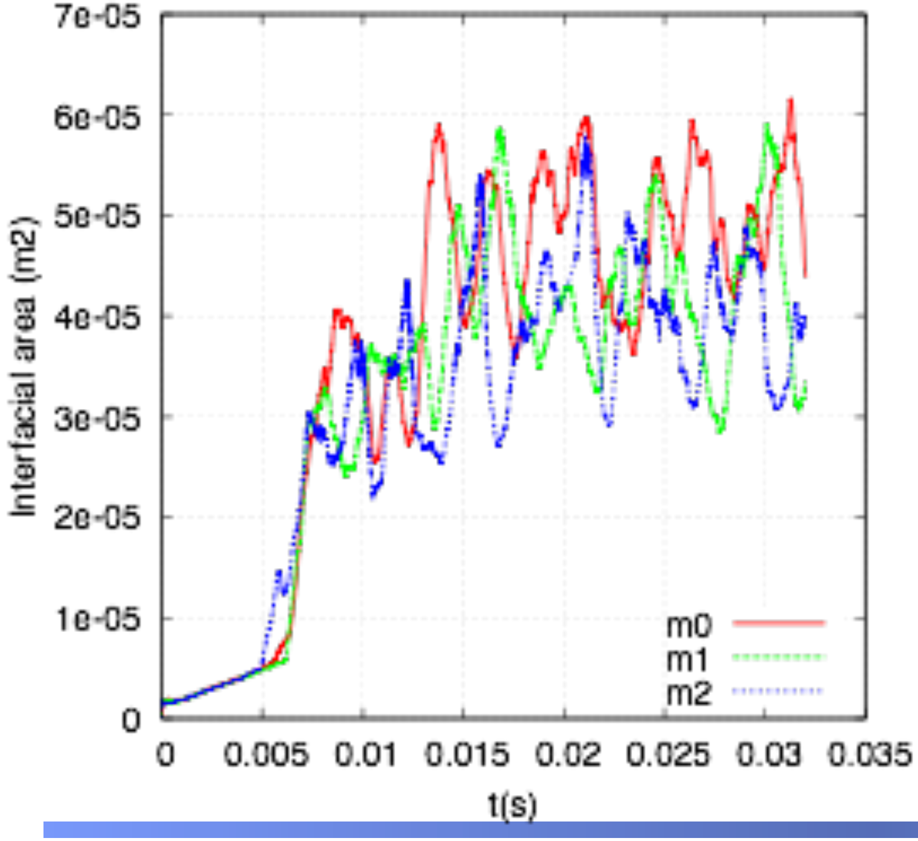


Liquid Kinetic Energ

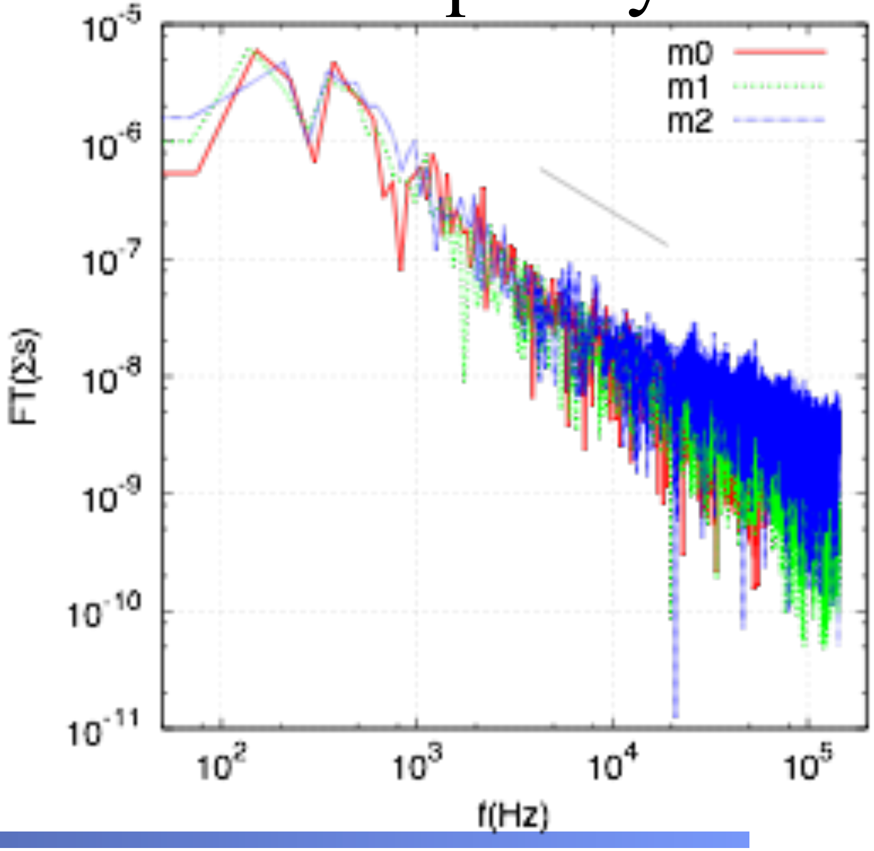


Total Interfacial Area

Time

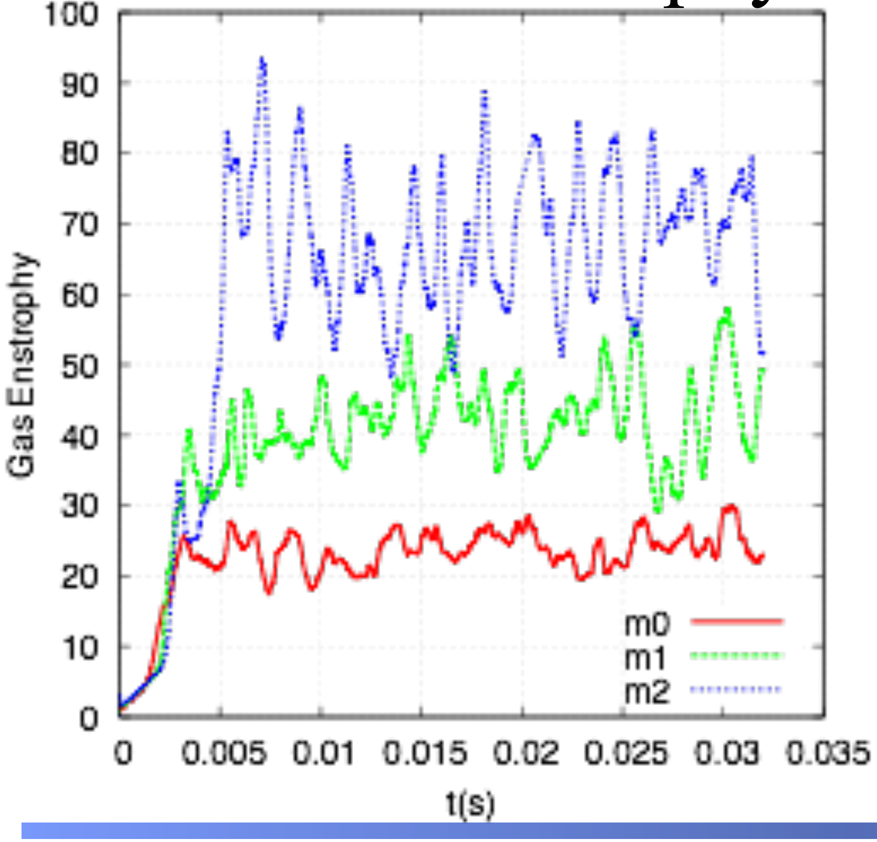


Frequency

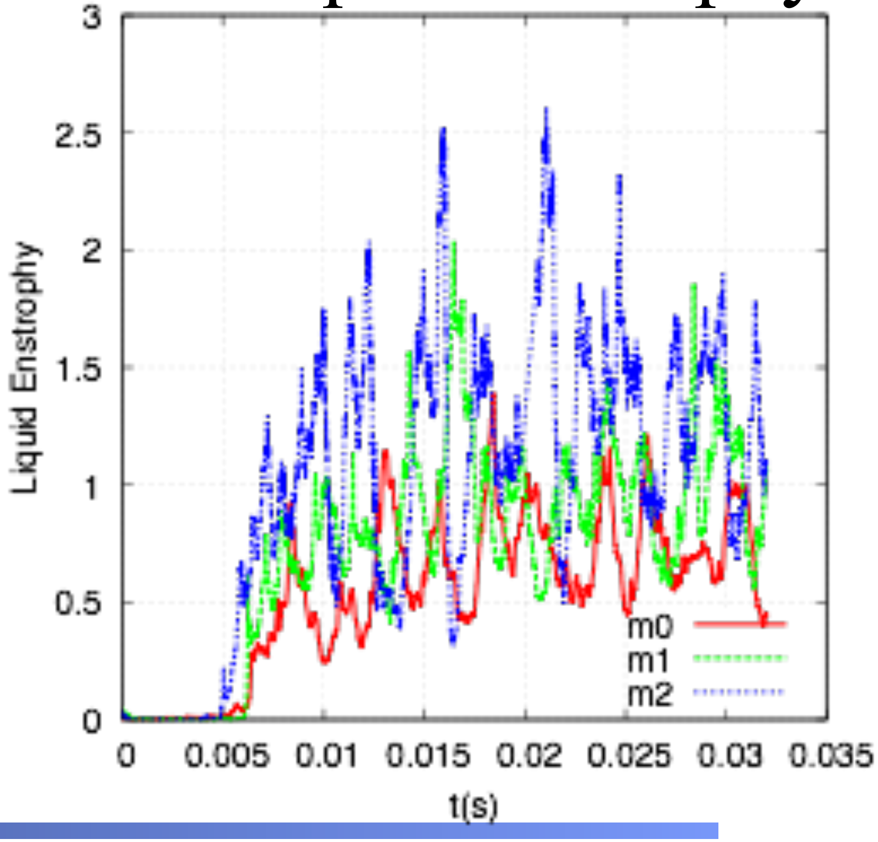


Enstrophy

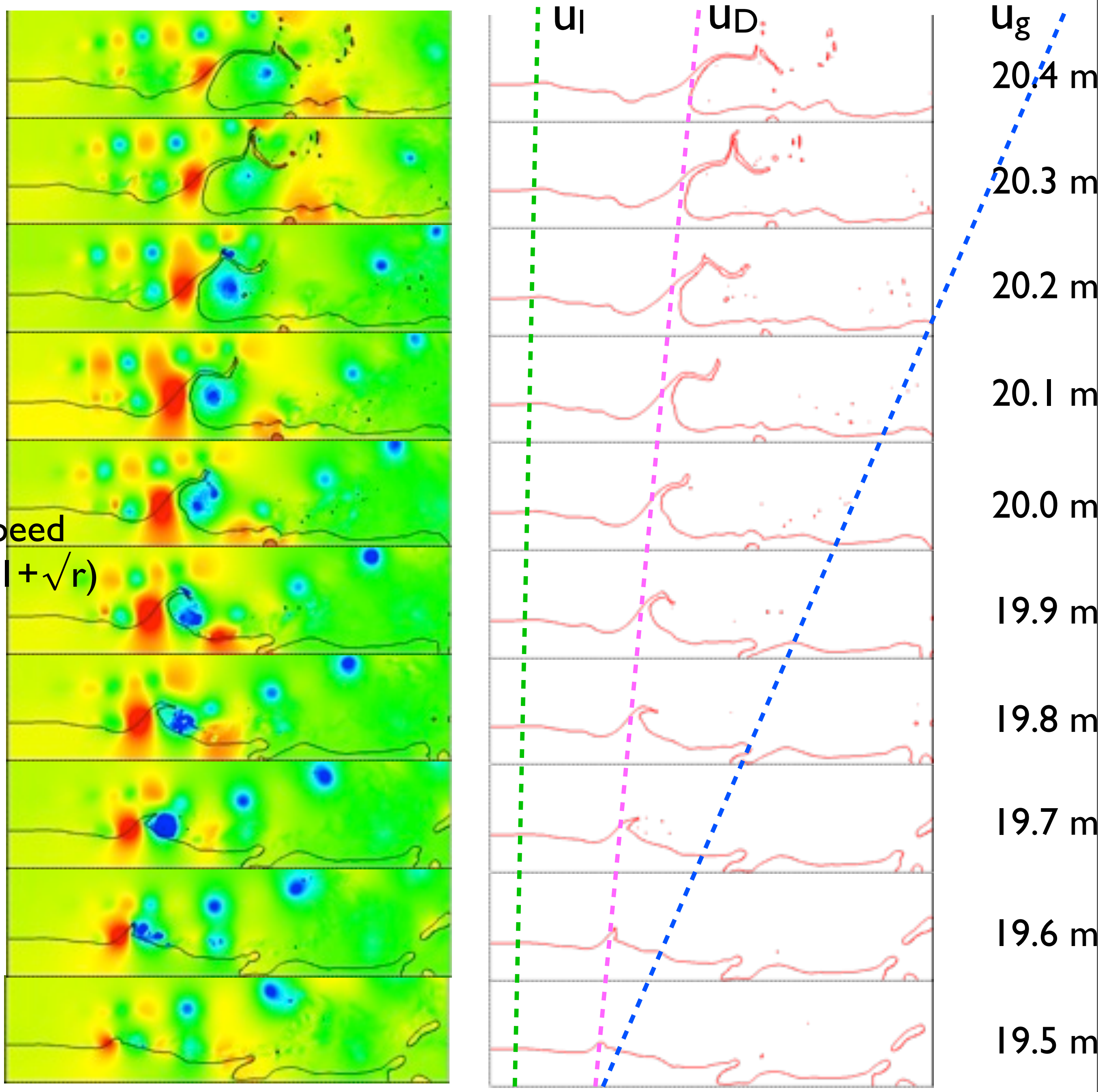
Gas Enstrophy



Liquid Enstrophy



Pressure & Interface Evolution

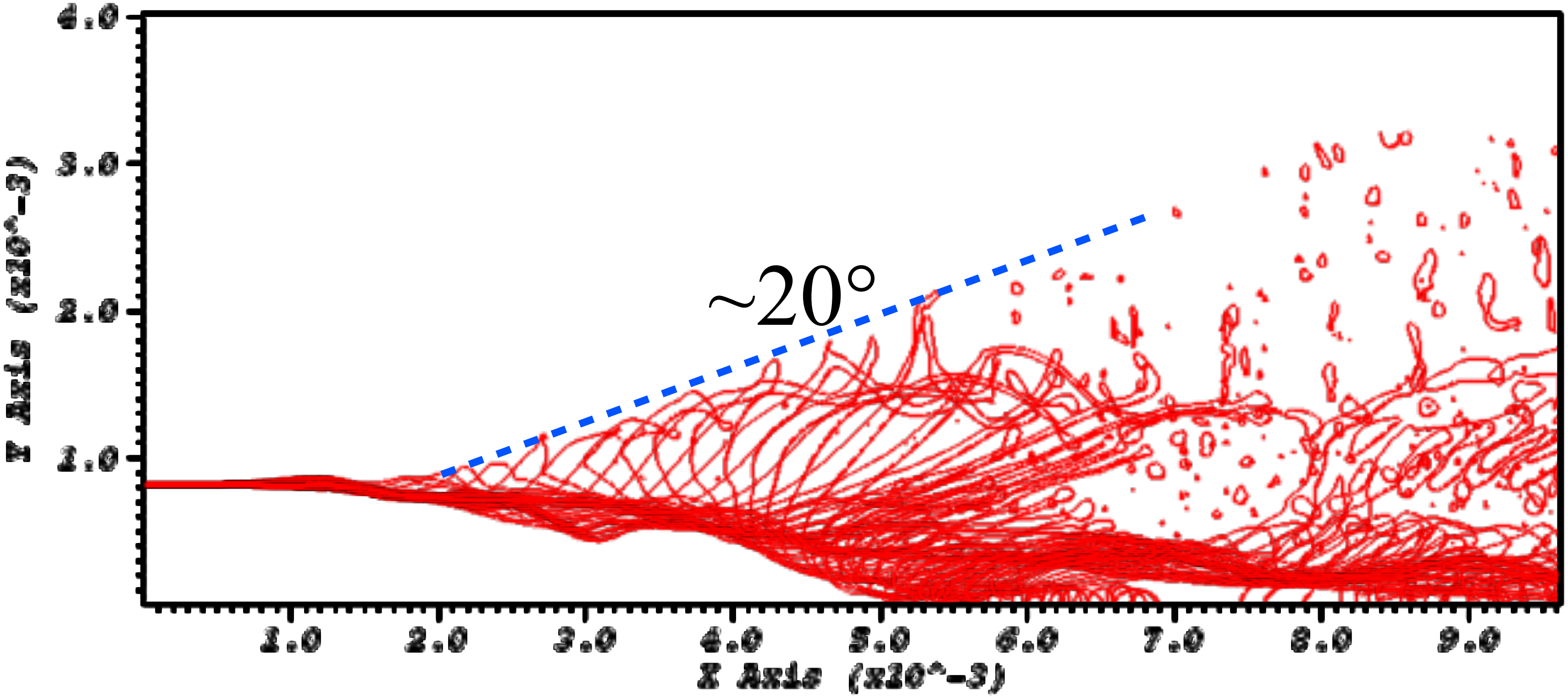


Dimotakis speed

$u_D=(u_l+\sqrt{r}u_g)/(1+\sqrt{r})$

Interfacial Wave Evolution

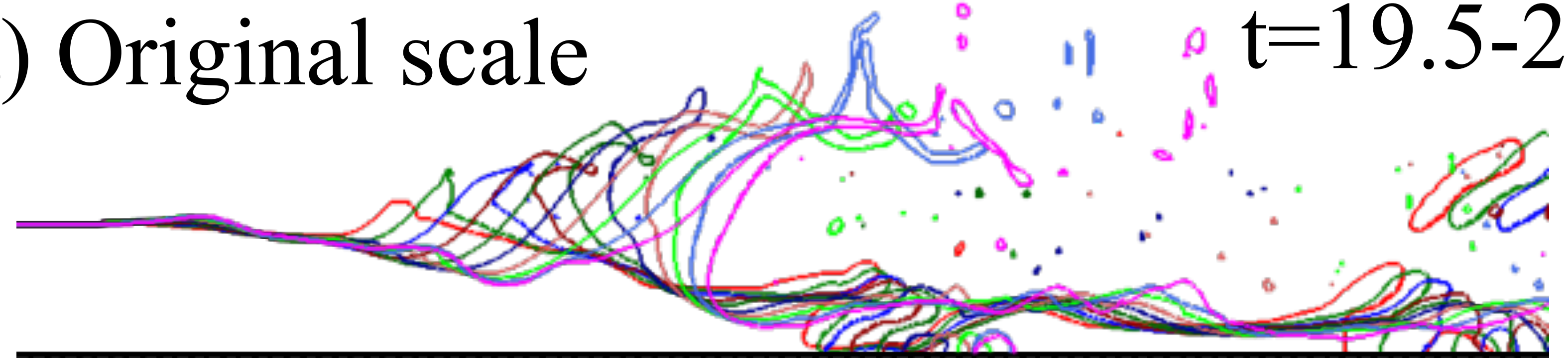
2D Slice of interface from t=19.0-21.9m



Self-Similar Wave

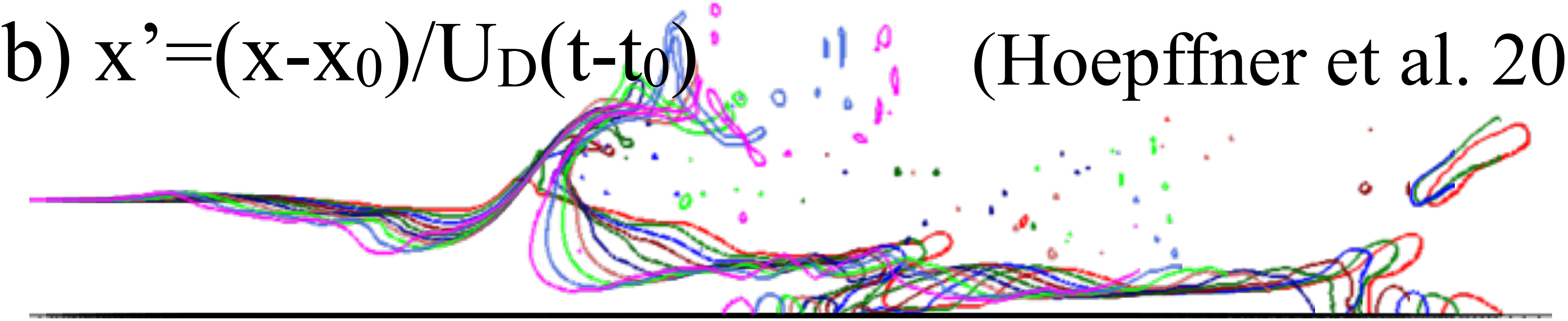
(a) Original scale

t=19.5-20.4 r



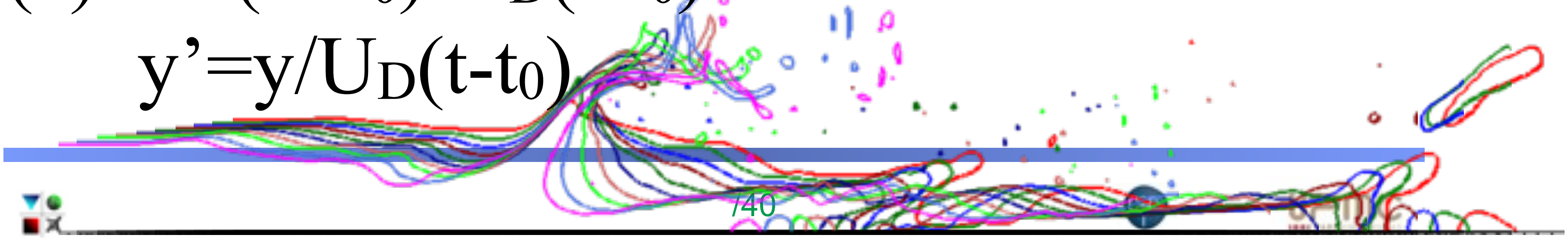
(b) $x'=(x-x_0)/U_D(t-t_0)$

(Hoepffner et al. 201



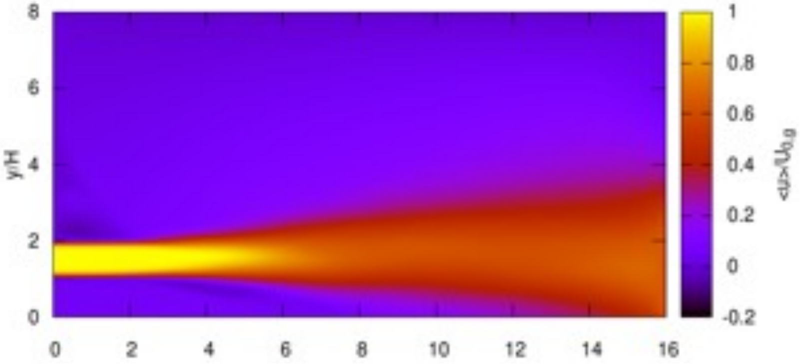
(c) $x'=(x-x_0)/U_D(t-t_0)$

$y'=y/U_D(t-t_0)$

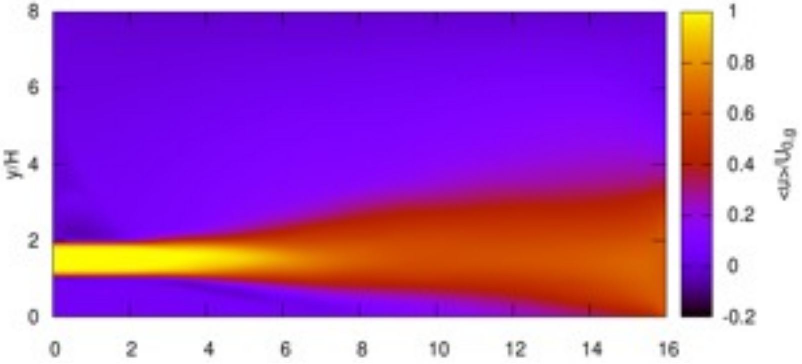


Averaging Velocity

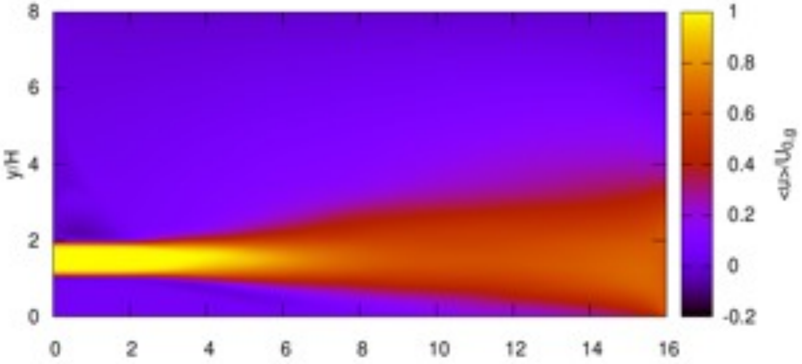
m0



m1

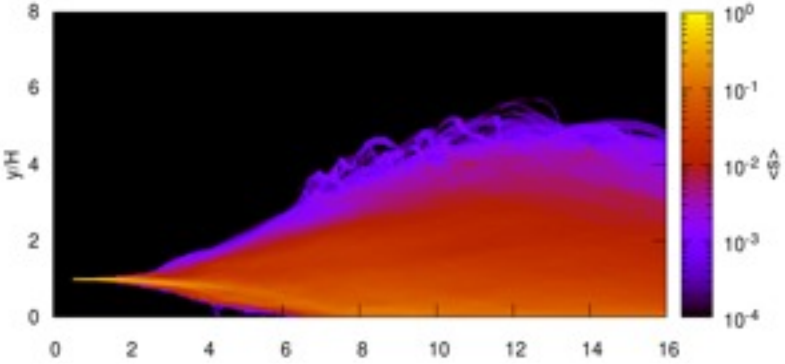


m2

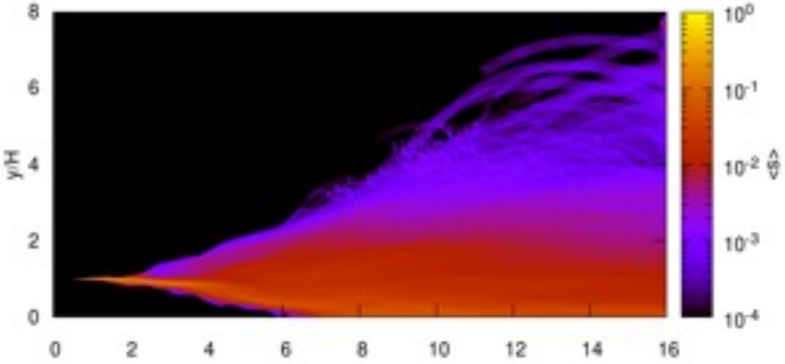


Averaging Interfacial Area

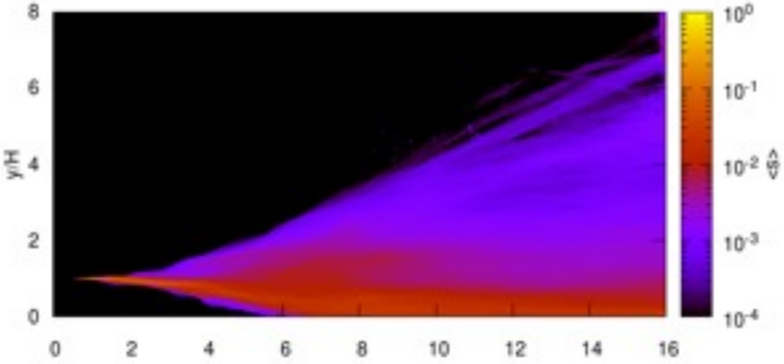
m0



m1

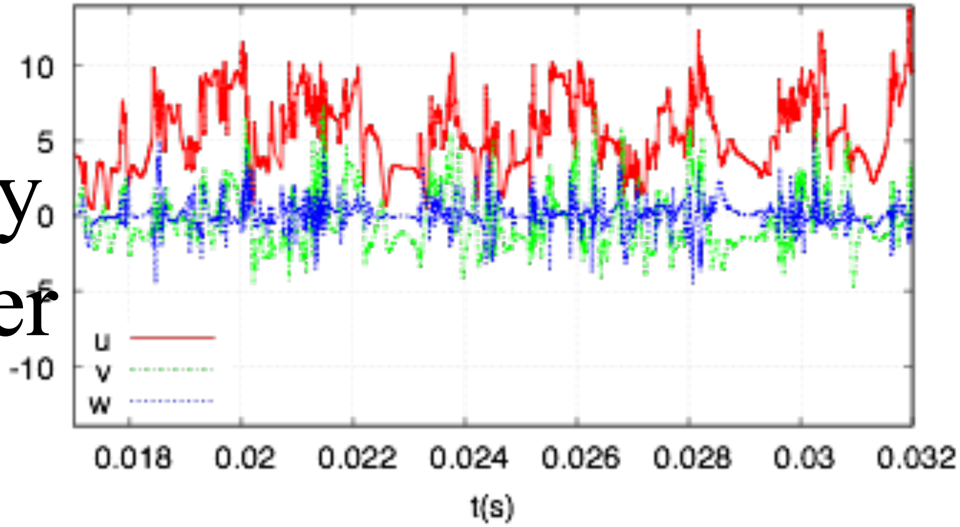


m2

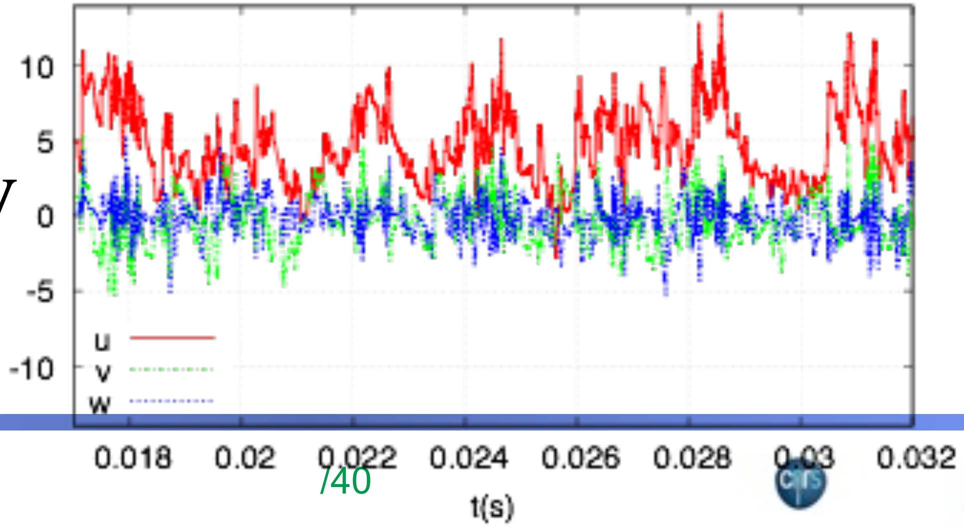


Turbulence Fluctuations

Top jet bdry
Mixing layer

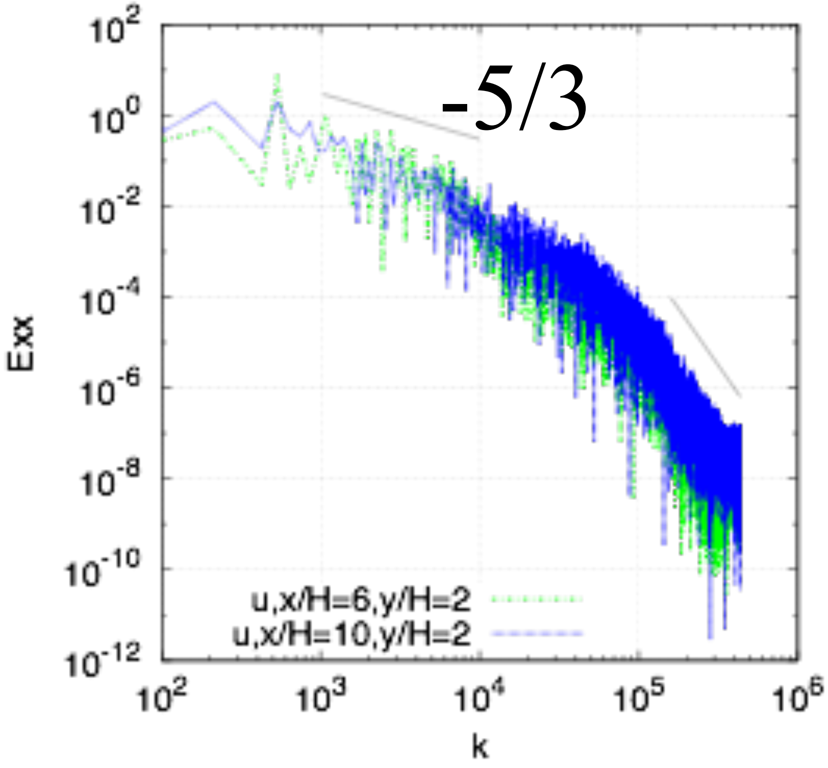


Low jet dry
Interface

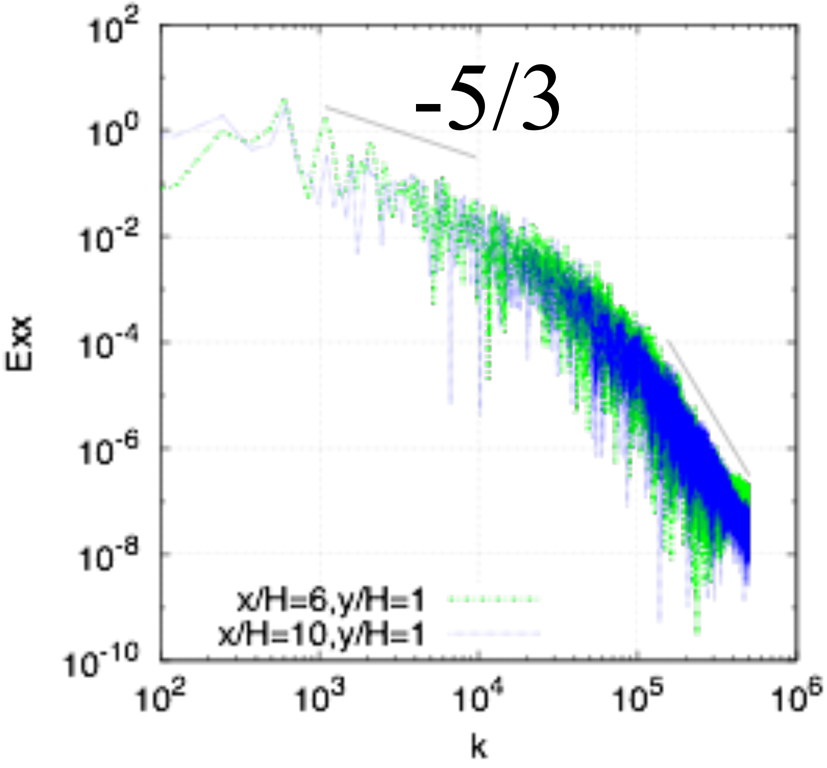


Energy Spetra

Gas-gas mixing layer

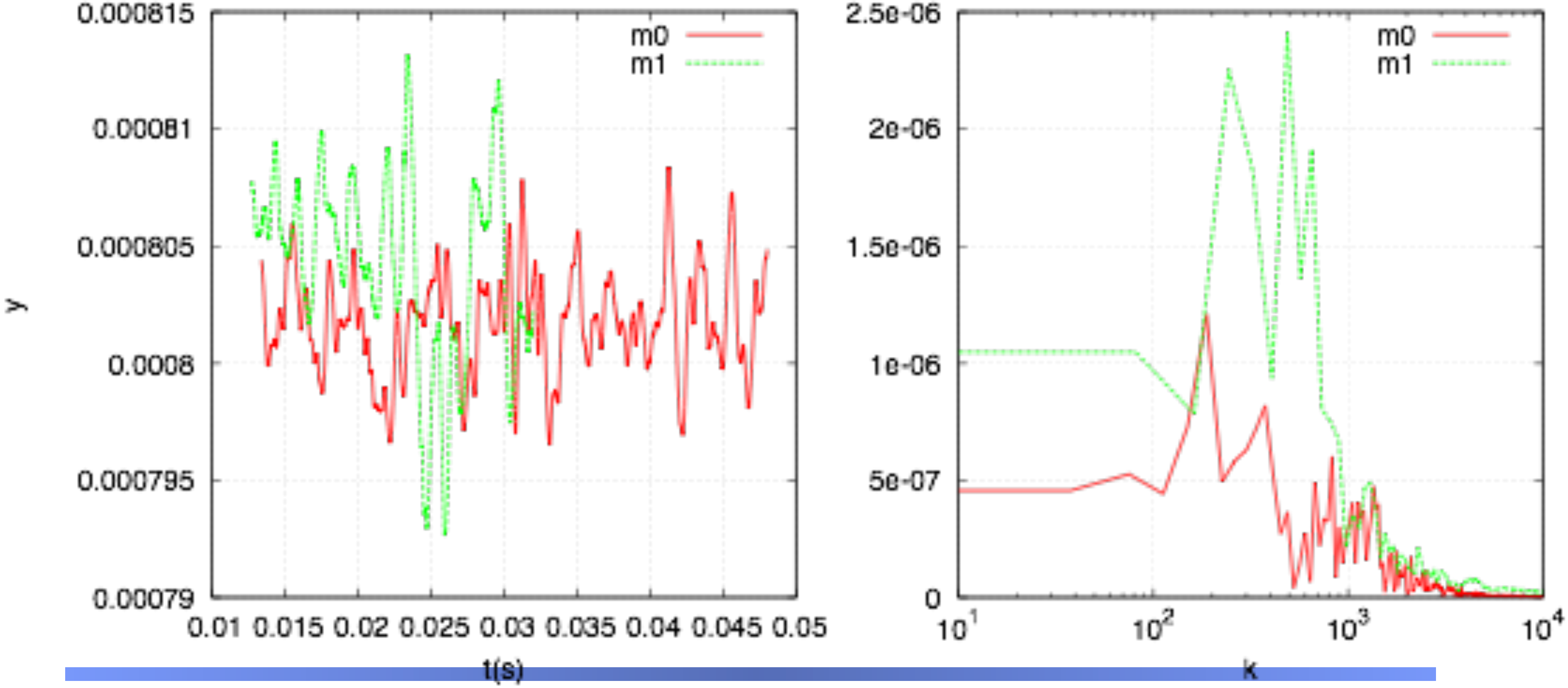


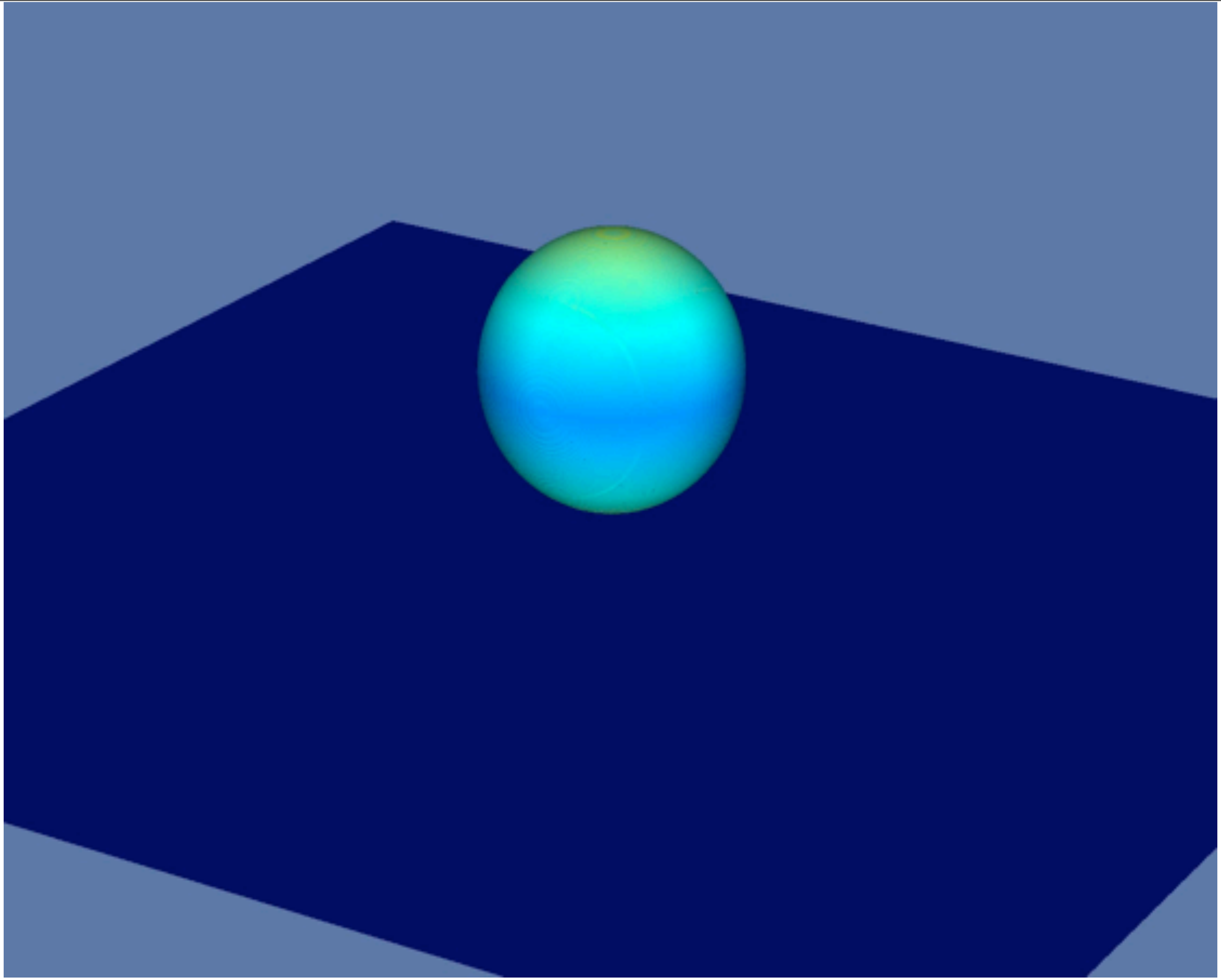
Gas-liquid mixing lay



Interfacial Instability

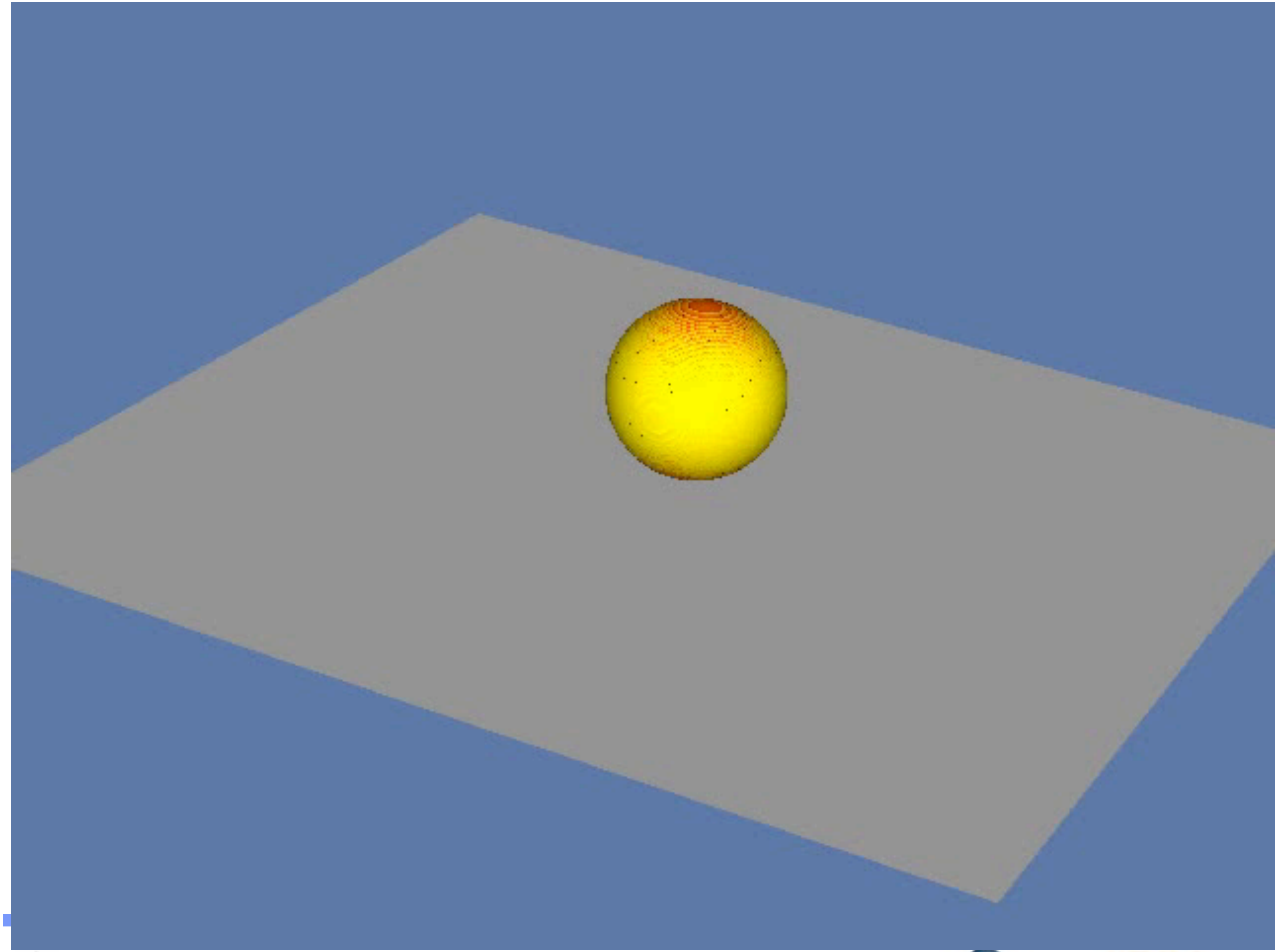
m0 m1





Same as before, higher resolution (Pascal Ray)

Same as before, higher resolution (Pascal Ray)



The iconic Marmottant-Villermaux case

Air-Water

$u_{liq} = 0.6 \text{ m s}^{-1}$, $u_{gas} = 35 \text{ m s}^{-1}$

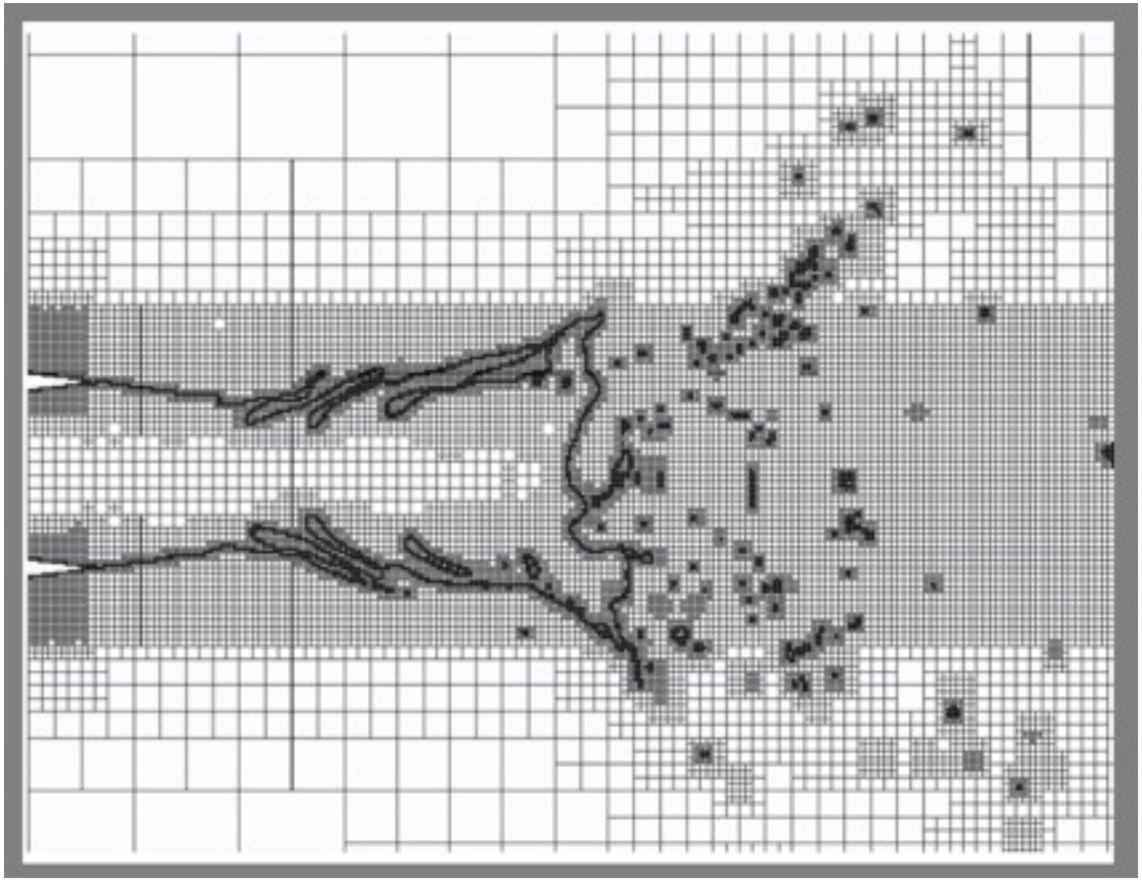
injection diameter D = 7.8 mm

$Re_g = 16000$, $We_g = 200$
based on D_1

Simulation : Two months on 64 AMD processors

line of eight 512^3 boxes – (equivalent regular mesh but we use octree adaptation)

Difficult to go to higher levels of refinement

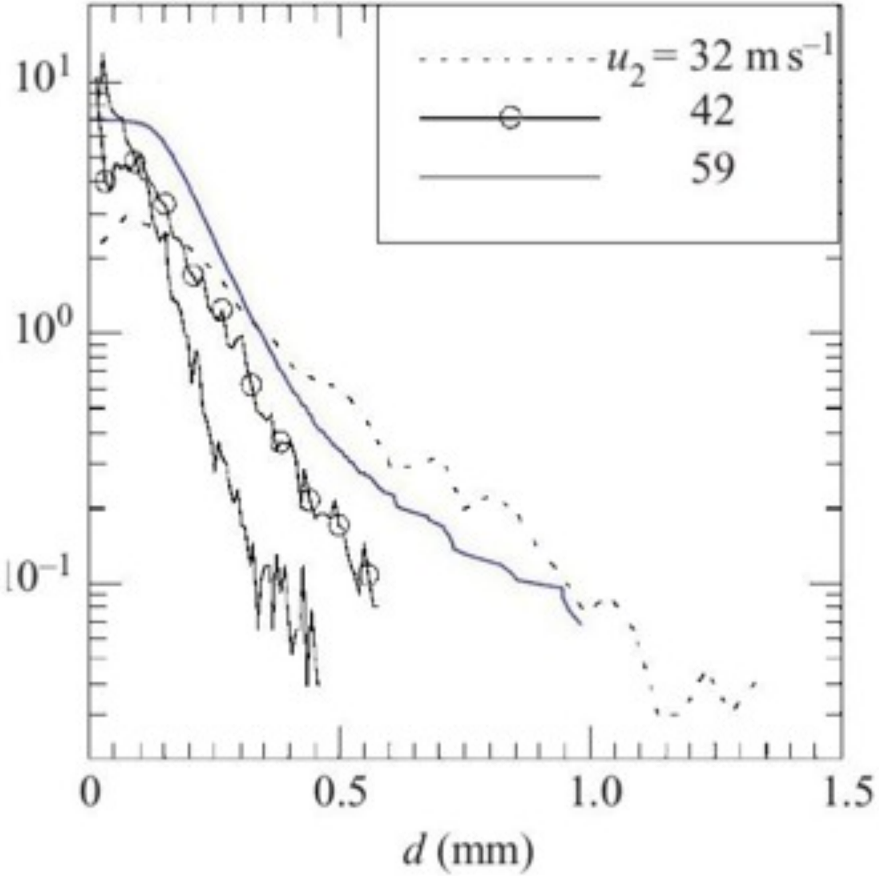


Boundary layer size
157 µm

Injector thickness
comparable.

Simulation: Gilles Agbaglah

Simulation: Gilles Agbaglah



Particle size distributions

experimental and computed
at UPMC

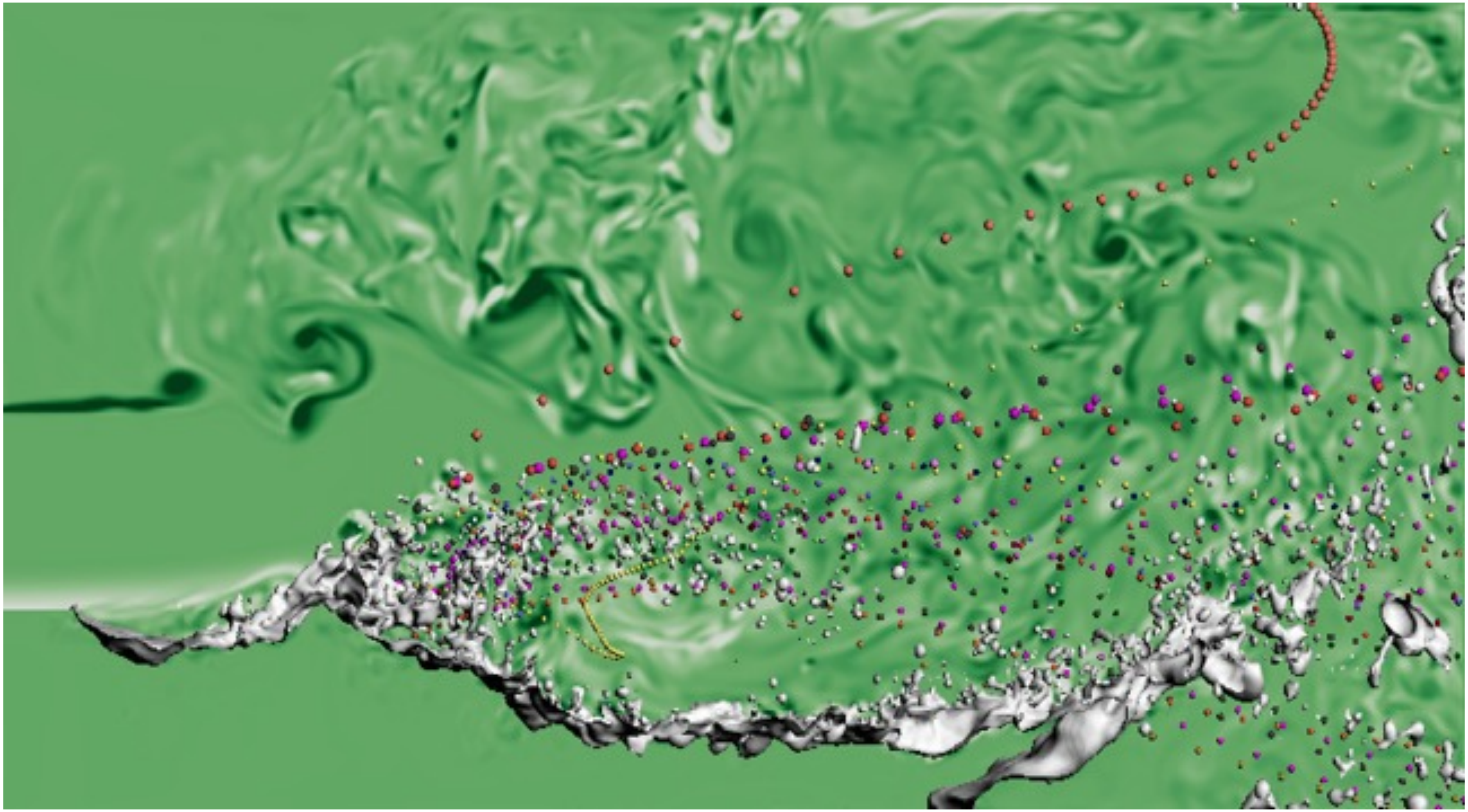
$u_2 = 35$ m / s

Planar shear flow of Descamps, Matas & Cartellier.
2nd colloque INCA, 2008.

Particle trajectories are measured.

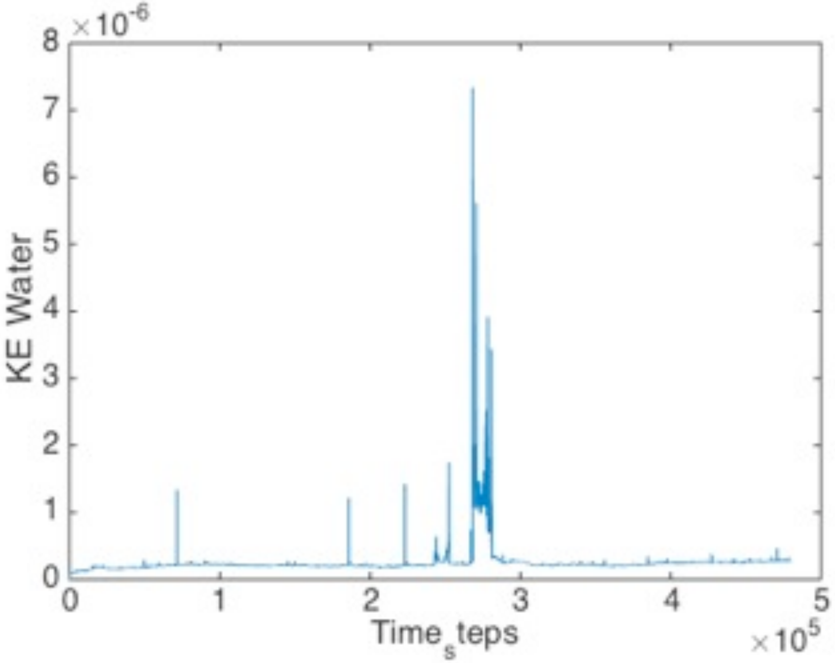
$Re_{\delta} = 1000$

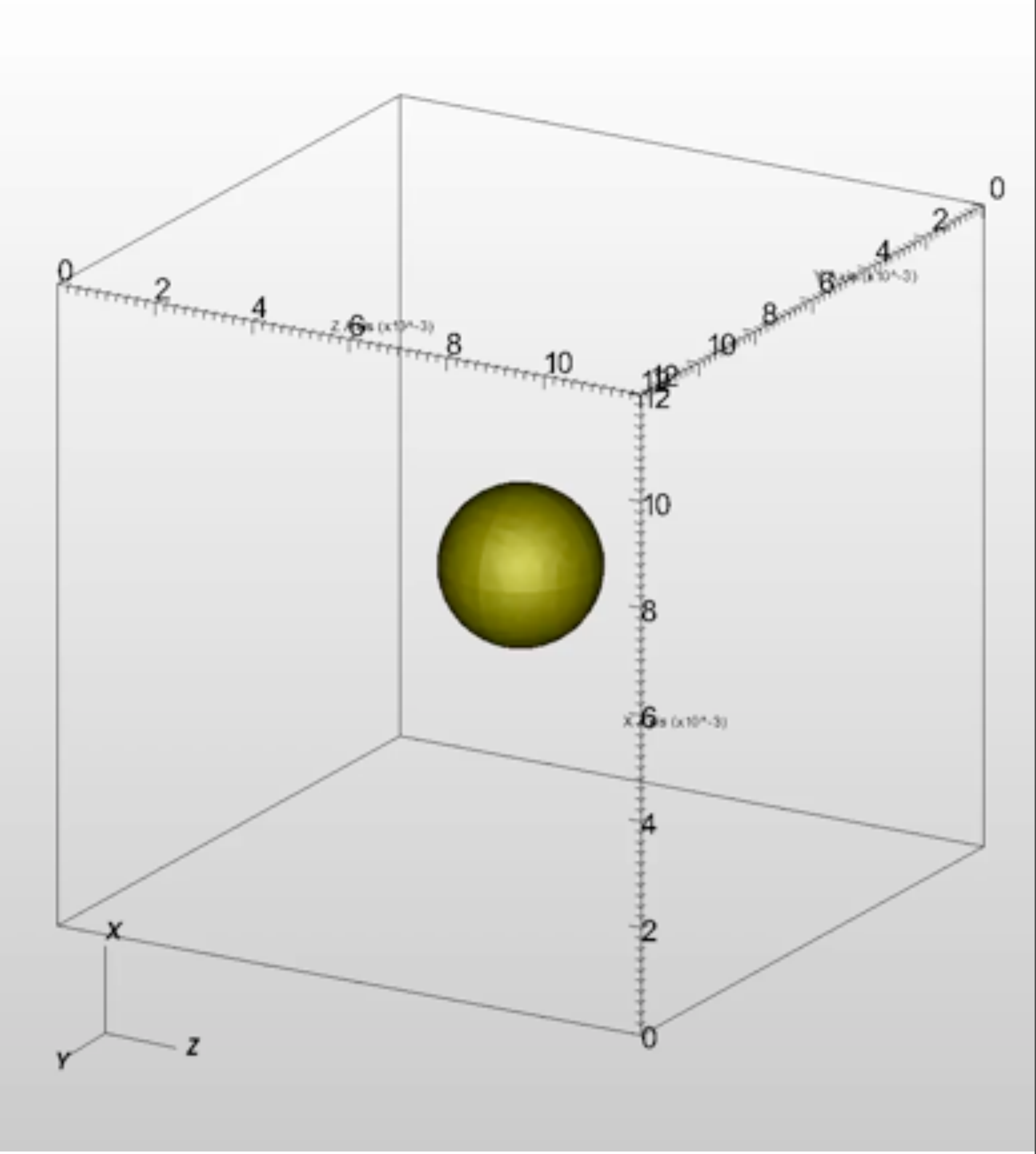
Simulation 64 x 256 x 512



Refinement level 1.D/Δx=15.
Spurious currents observed.

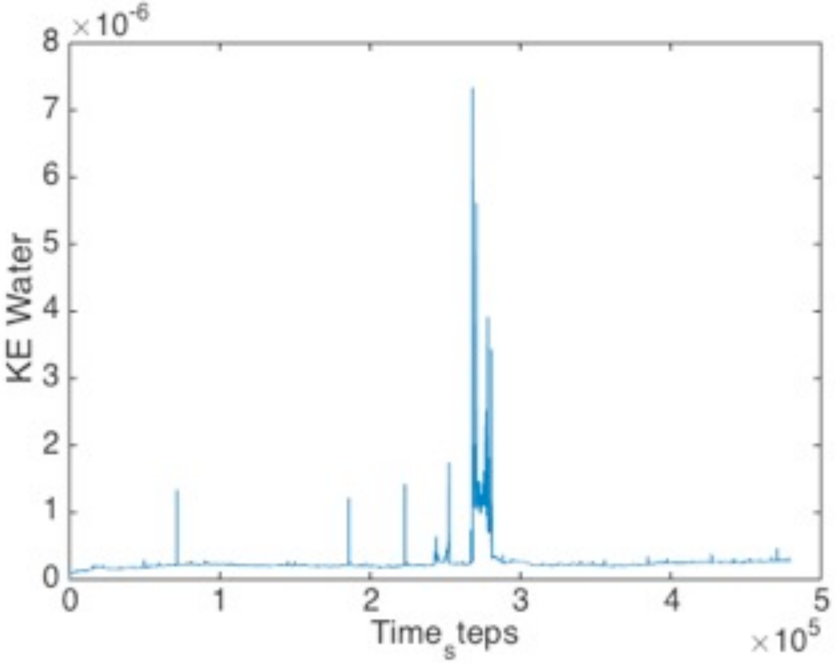
Kinetic energy of the droplet



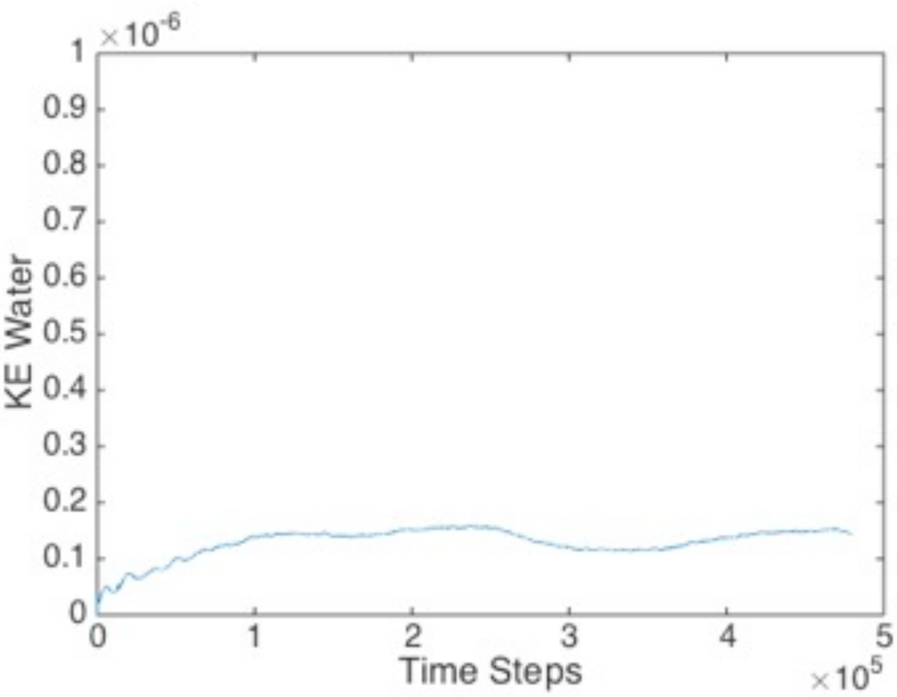


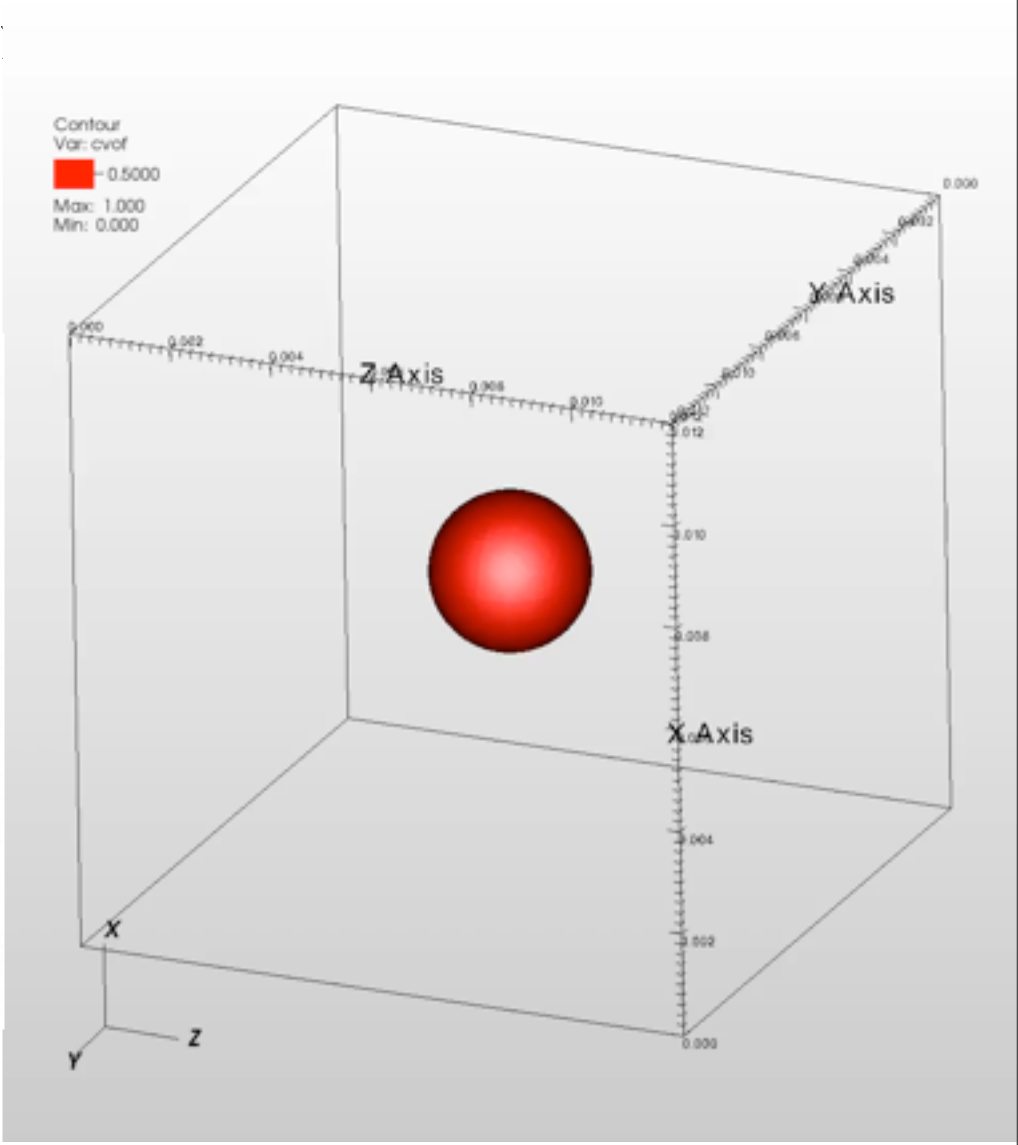
Refinement level 1.D/Δx=15.
Spurious currents observed.

Kinetic energy of the droplet

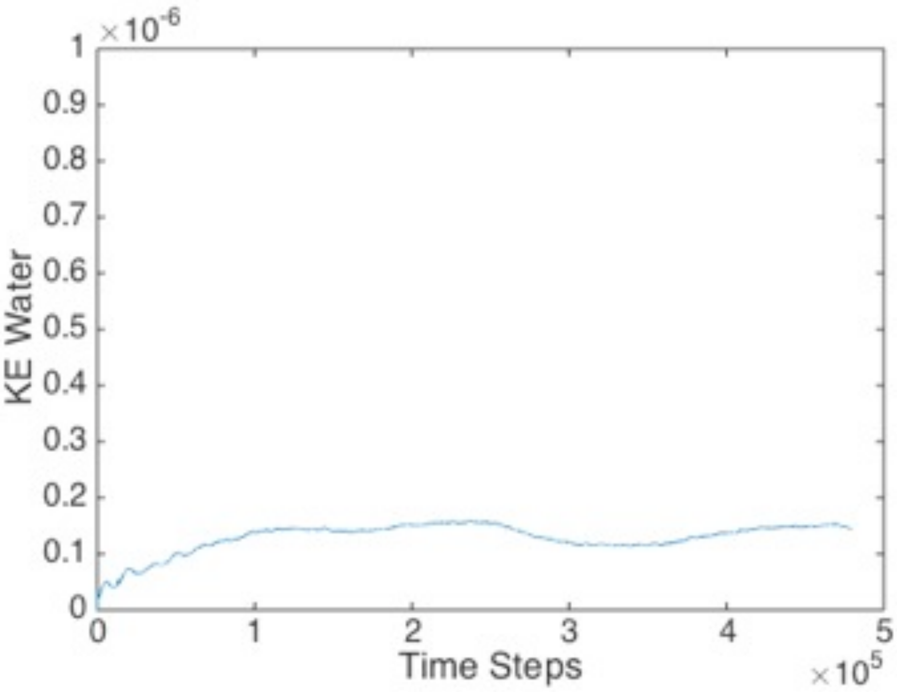


Refinement level 2. D/Δx=30.
Kinetic energ

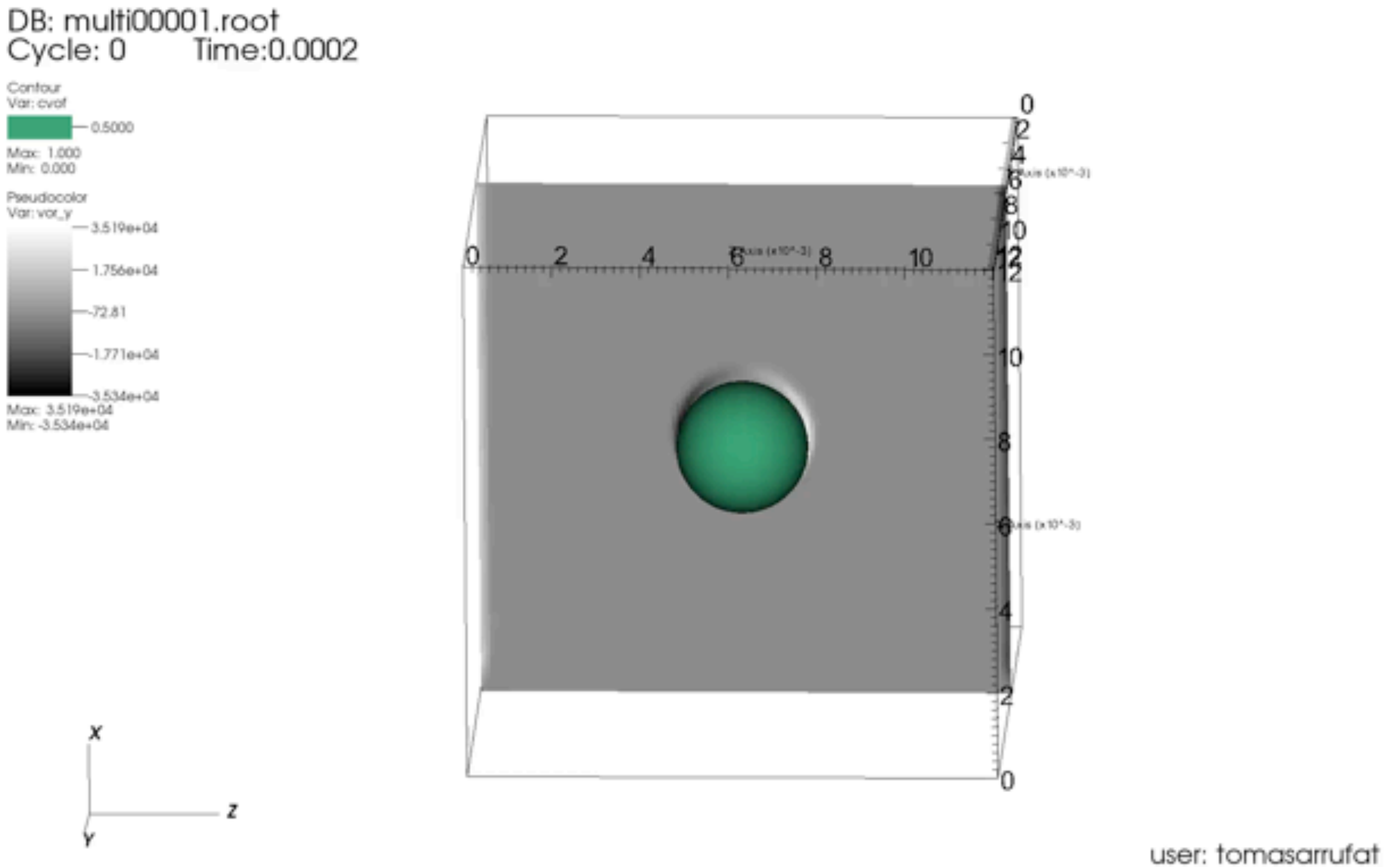




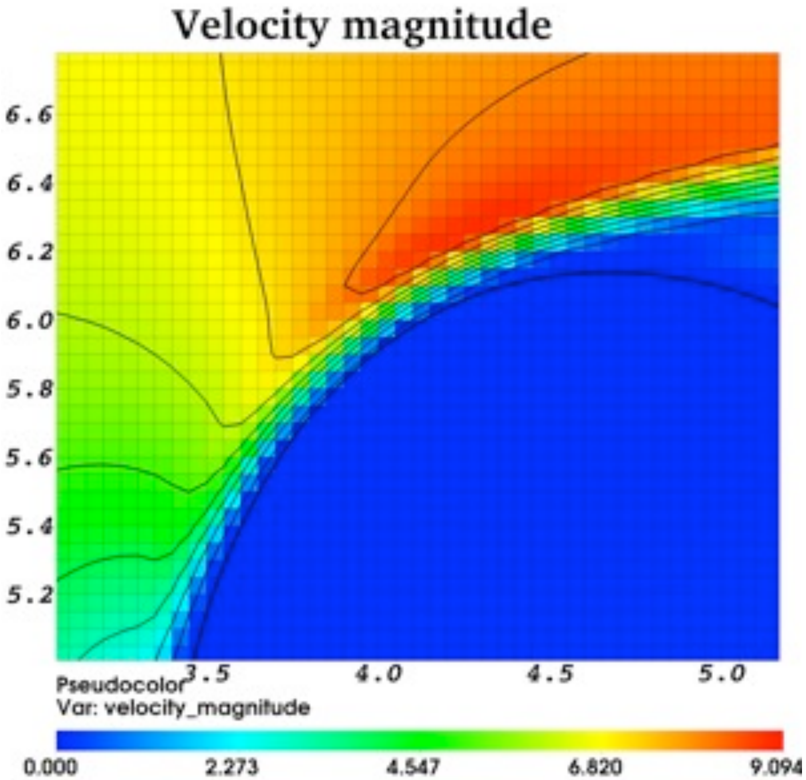
Refinement level 2. D/Δx=30.
Kinetic energy of the droplet

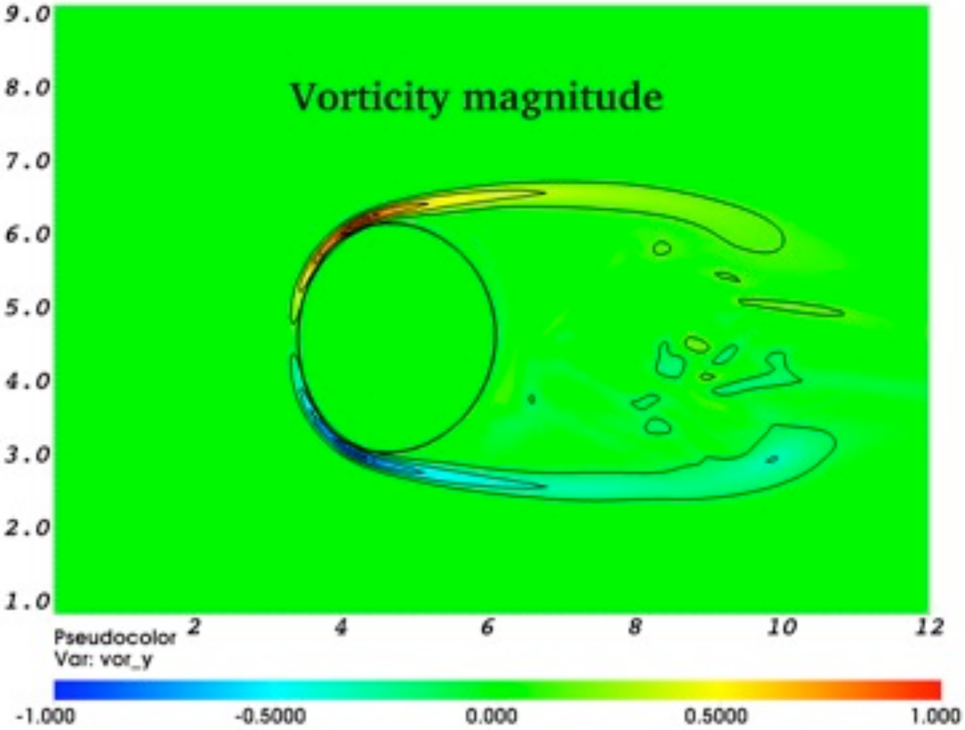


diameter d=8 mm



diameter d=8 mm



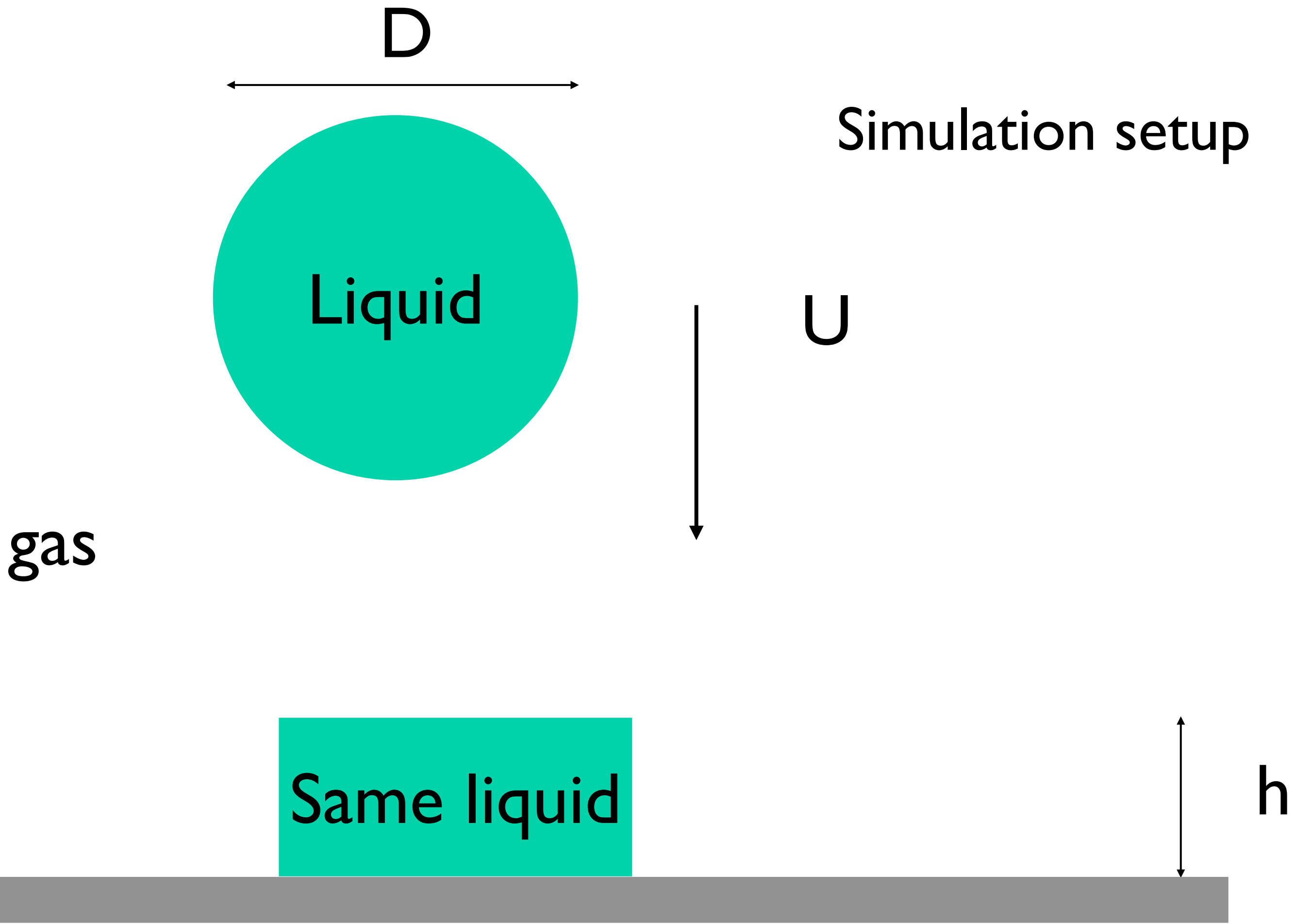


- Even for D/Δx = 60 the boundary layer is only covered by 5 cells.
- The boundary layer is very small relative to the droplet diameter.
- Such results suggest that the accurate solution of air flow with water droplets can be extremely challenging.

dálembert

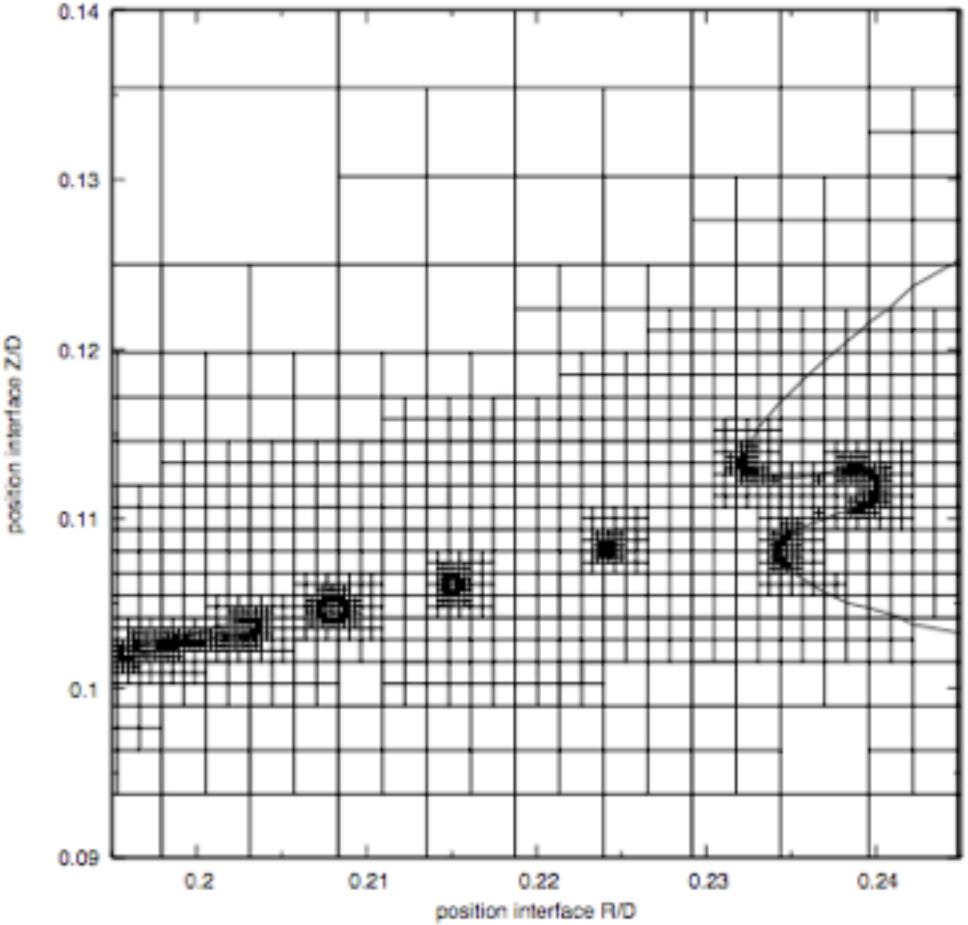
DROPLET IMPACT

Simulation setup



experiments by Thoroddsen

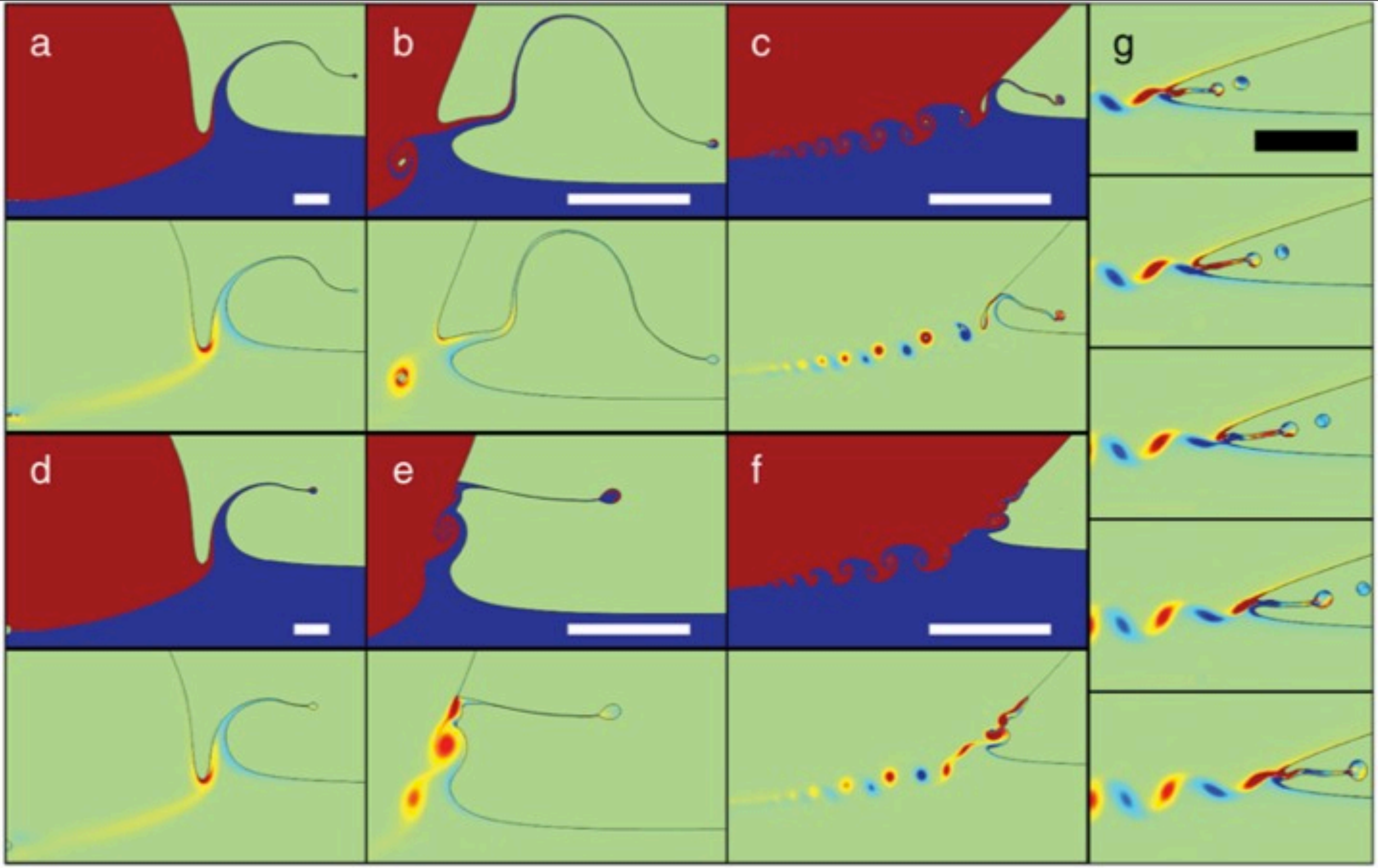
Even higher resolutions, with octree-AMR, up to D/h ~ 6000.



From Josserand, Ray and SZ, ICMF2010

Simulation Pascal Ray
Gerris code

Adapting on curvature and (less strongly) on vorticity



Thoroddsen, Thoraval & others + Gerris, KAUST, Phys. Rev. Lett. (2012)

Figure 4f

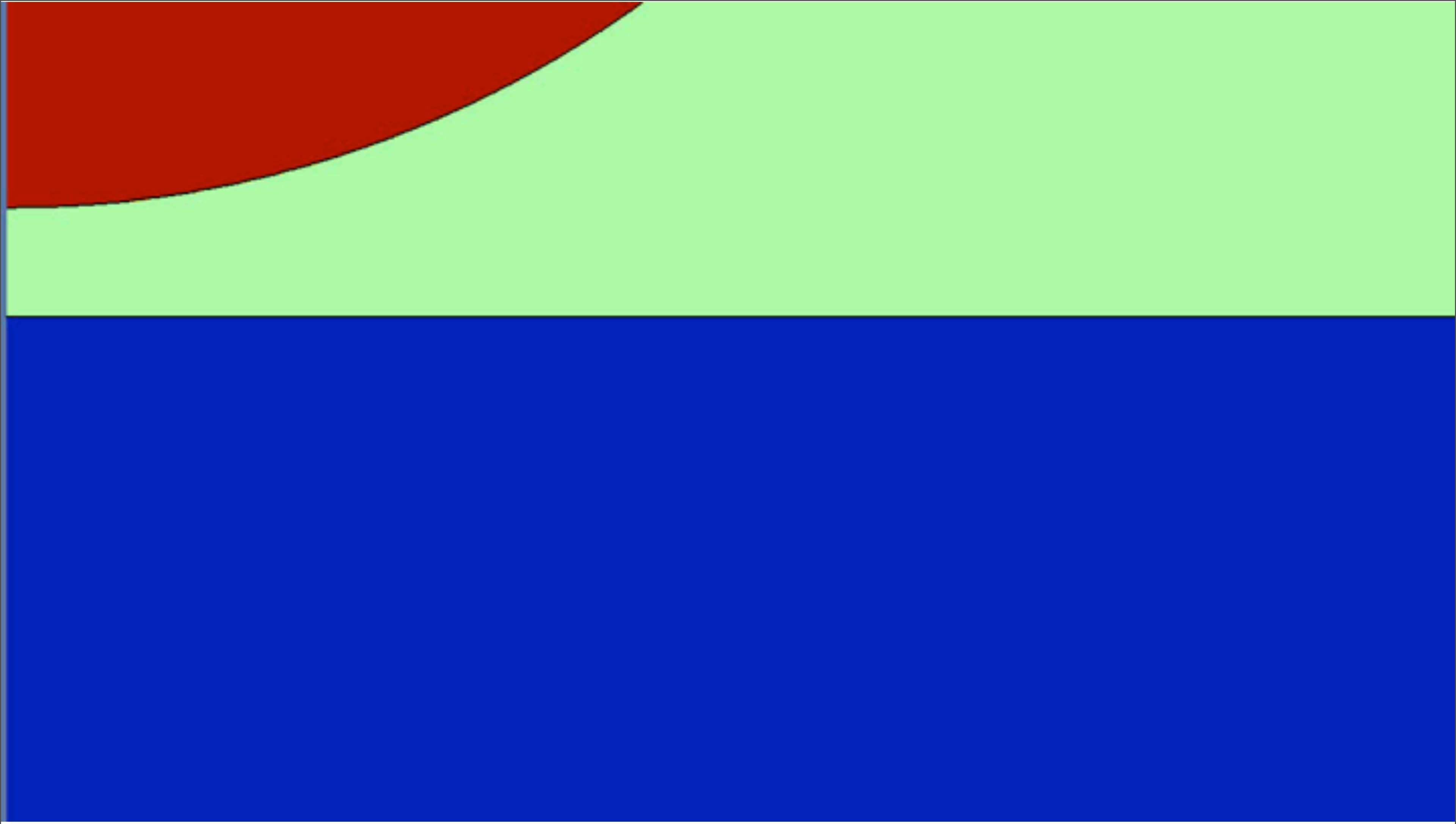


Figure 4f