オープンCAEシンポジウム2014(東京) @国立オリンピック記念青少年センター
2014年11月13日(木)～14日(金)

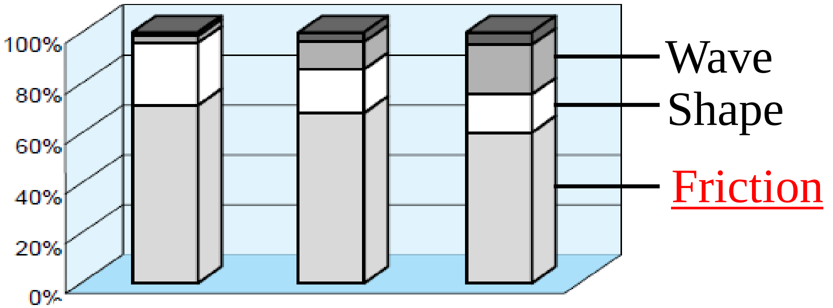
OSAKA UNIVERSITY

channelFoamを用いた チャネル乱流の直接数値計算

○ 中本 真義，高木 洋平, 岡野 泰則
大阪大学基礎工学研究科

Introduction

Drag reduction for energy saving of ship



Friction coefficient : $C_T = C_S$ (Shape) + C_W (Wave) + C_f (Surface Friction)

Drag reduction techniques

Commercial hydrogel paintings
LF-Sea (Nippon Paint)
Hydrogel formation
DR ~ 10%
Biomimetic
Water trapping effect

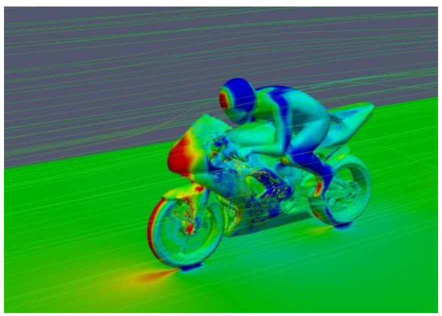
Numerical Simulation of wall turbulence

OSAKA UNIVERSITY

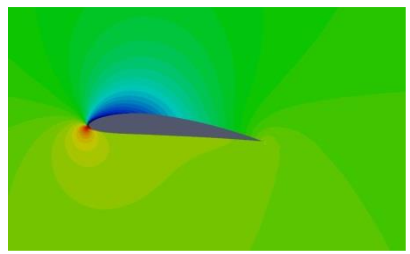
channelFoam ··· incompressible solver for channel flow

Complicated geometry

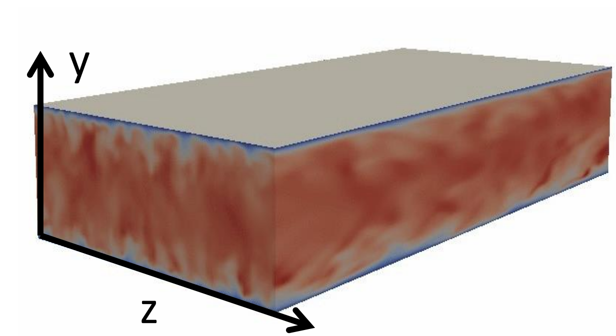
Motor bike case



Wing case



Simple geometry

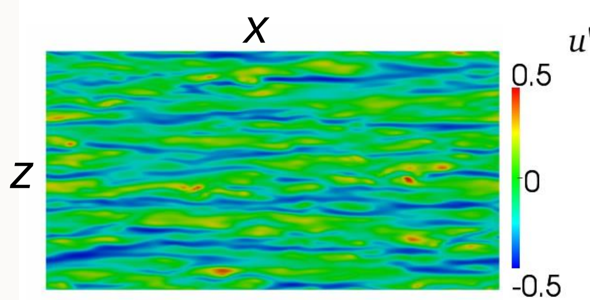


Vortices

Q : standard utility

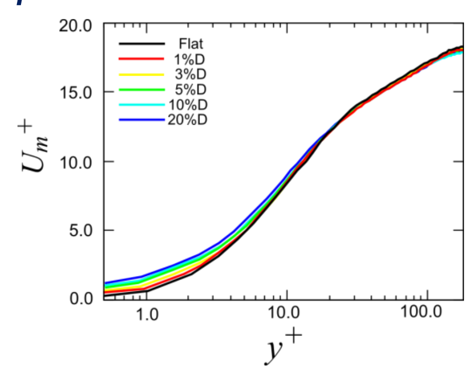
Streaks

Velocity fluctuation :



Turbulent statistics

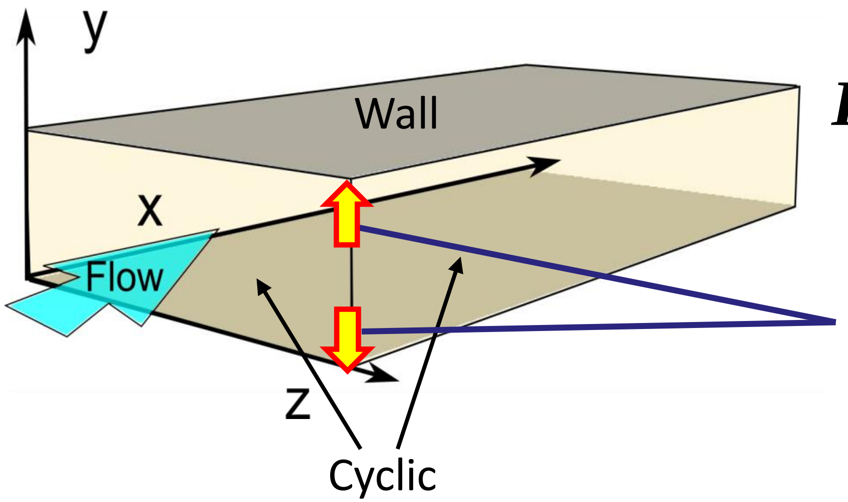
postChannel



Code validation of channelFoam

Code validation

LESModel : *laminar* (No turbulence model)



$$\boldsymbol{Re} = \frac{2u_m\delta}{\nu} = 5600$$

u_m: mean velocity
δ : channel-half width
ν : kinematic viscosity

Label	Grid number	Method
Kim et al. (1987)[1]	192×129×160	Fourier–spectral
Present study	128×128×128	FVM with OF

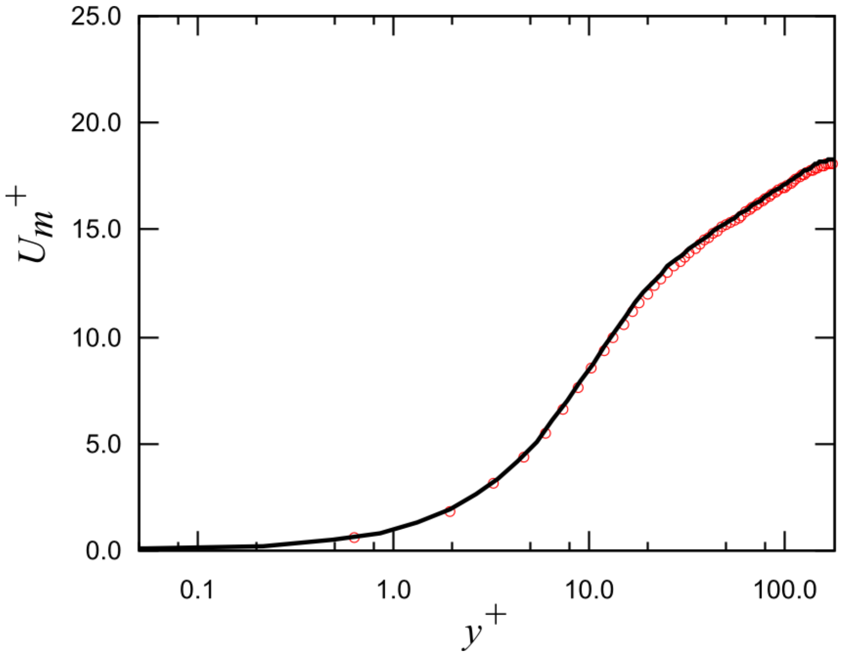
[1] J. Kim *et al., J. Fluid Mech.*, **177**, pp. 133-166 (1987).

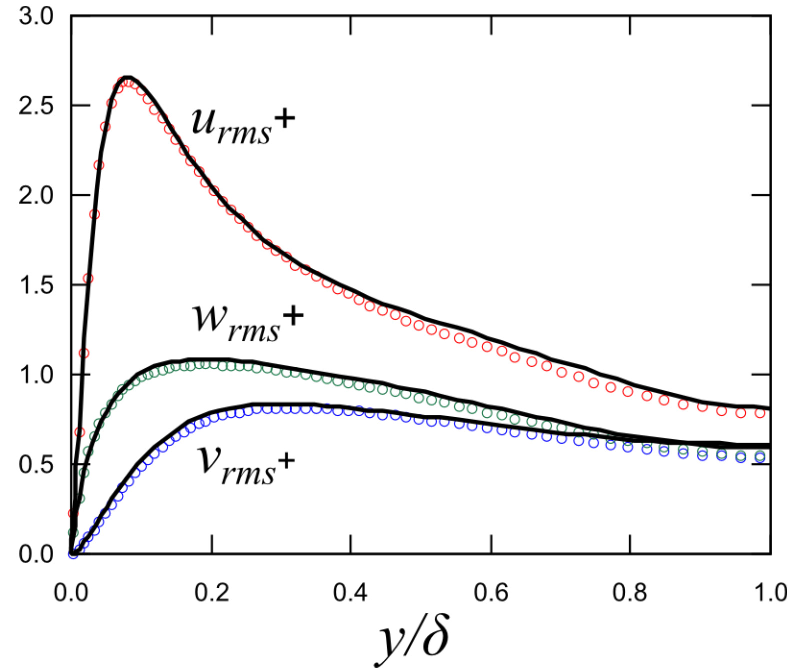
Code validation

Friction coefficient : $C_f = \dfrac{\tau_w}{\rho u_\tau^2}$

Kim et al. (1987) $C_f = 8.18 \times 10^{-3}$

Present study $C_f = 8.07 \times 10^{-3}$





The relative error of friction coefficient was 1.0%

Numerical simulation by channelFoam

- Turbulent channel bounded with wavy walls

- Turbulent channel bounded with porous walls

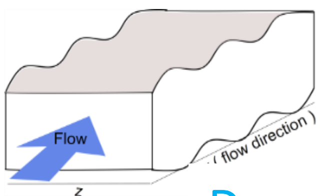
Turbulent channel bounded with wavy walls

$$DR = \frac{\left(-\left.\frac{dP}{dx}\right|_{flat}\right)-\left(-\left.\frac{dP}{dx}\right|_{wavy}\right)}{\left(-\left.\frac{dP}{dx}\right|_{flat}\right)}$$

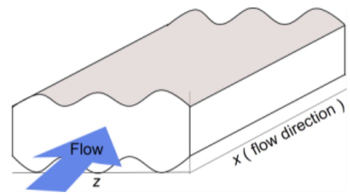
$-\left.\frac{dP}{dx}\right|_{flat}$: mean pressure gradient in a flat channel

$-\left.\frac{dP}{dx}\right|_{wavy}$: mean pressure gradient in a wavy channel

Case 1



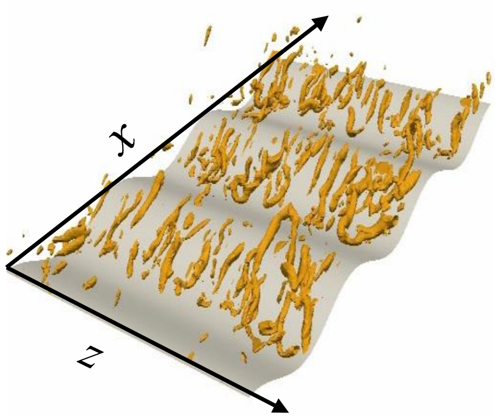
Case 2



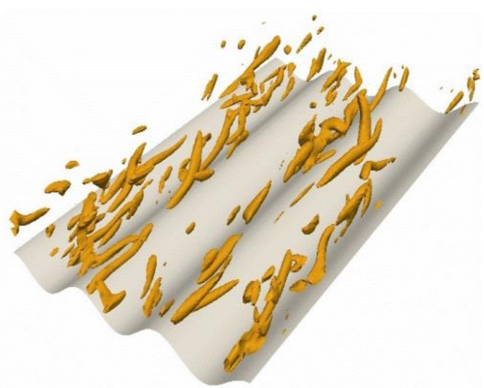
Case 3

Drag increase

DR = -20.4 %

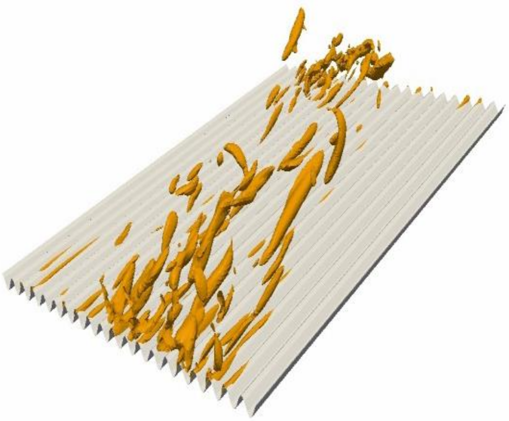


DR = -9.4 %



Drag reduction

DR = +9.3 %



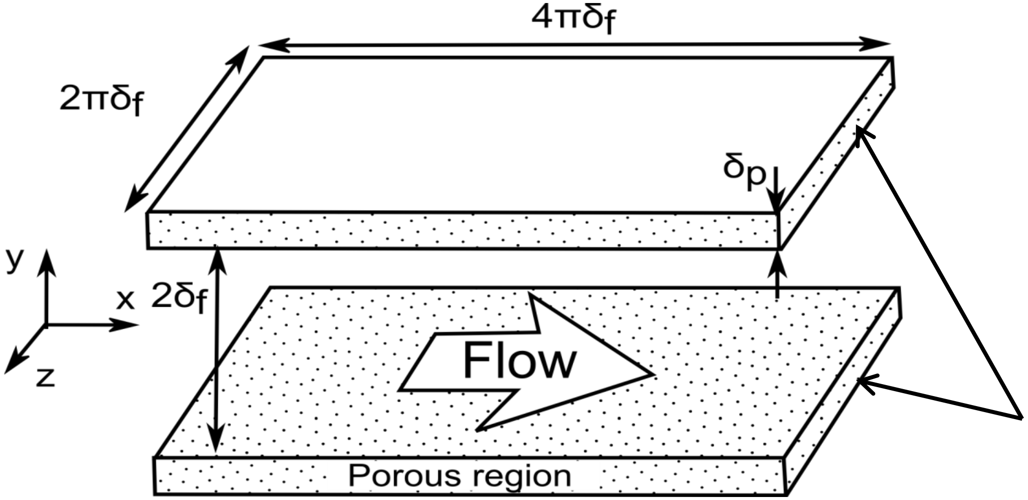
Numerical simulation by channelFoam

- Turbulent channel bounded with **wavy** walls

- Turbulent channel bounded with **porous** walls

Numerical simulation for turbulent channel flow bounded with porous walls

Developped by porous***Foam



$$Re_m = \frac{2u_m(\delta_p+\delta_f)}{\nu} = 5600$$

δ_f : Half-width of fluid region

δ_p : Porous wall thickness

cellZone : porosity

Case	1%D	3%D	5%D	10%D	20%D	Flat
Porous wall thickness δ_p	$0.01\delta_f$	$0.03\delta_f$	$0.05\delta_f$	$0.10\delta_f$	$0.20\delta_f$	—
Total channel width	$2.02\delta_f$	$2.06\delta_f$	$2.10\delta_f$	$2.2\delta_f$	$2.4\delta_f$	$2.0\delta_f$
Number of grid points $N_x, (N_y^f + N_y^p), N_z$	149, (129+10), 129	149, (129+16), 129	149, (129+16), 129	149, (129+20), 129	149, (129+20), 129	129,129,129

DNS of channel flow with porous wall

OSAKA UNIVERSITY

Governing equations

Continuity equation

$$\nabla \cdot \boldsymbol{u} = 0$$

Momentum equation

$$\frac{D\boldsymbol{u}}{Dt} = -\nabla p' + \nu \nabla^2 \boldsymbol{u} + (1-\varphi)\boldsymbol{S}$$

Numerical scheme

- **Spatial discretization scheme**
  Finite Volume Method
- **Time advancement**
  Implicit Euler method (1 st)
- **Poisson equation**
  PISO algorithm

Porous model

Darcy-Forchheimer equation

$$\boldsymbol{S} = -\frac{\nu}{K}\boldsymbol{u}_m - 2\frac{c_f}{\sqrt{K}}|\boldsymbol{u}_m|\boldsymbol{u}_m$$

φ	: Porosity	0.80
K	: Permeability	0.02 [mm^2]
c_f	: Forcheheimer coefficient	0.17

cf. K. Suga *et al* Int. J. *Heat Fluid Flow* (2011)

Results – Drag reduction Ratio

Case	δ_p^+	$C_f \times 10^3$	DR [%]
1%D	2	7.23	+0.3
3%D	5	6.47	+10.6
5%D	9	6.55	+9.6
10%D	18	7.16	+1.2
20%D	37	7.56	-4.3
Flat	-	7.25	-

Drag reduction Ratio

$$DR\ [\%] = \left(1 - \frac{C_f^p}{C_f^0}\right) \times 100$$

C_f^p:Friction coefficient of porous wall

C_f^0:Friction coefficient of flat wall

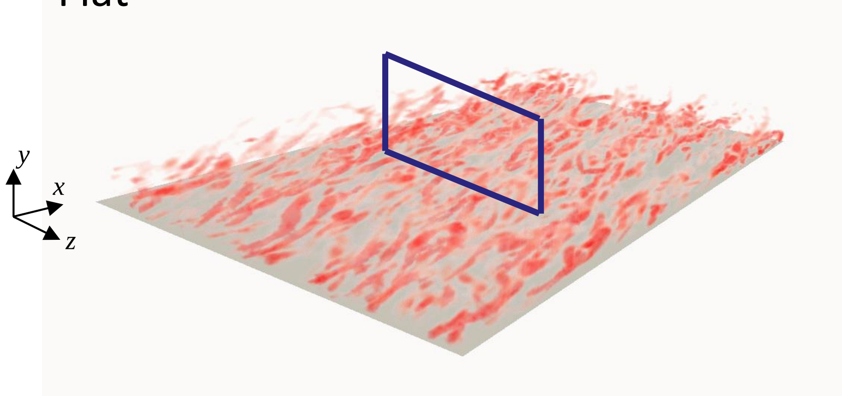
FIK identity for fully developed turbulent channel flow[1]

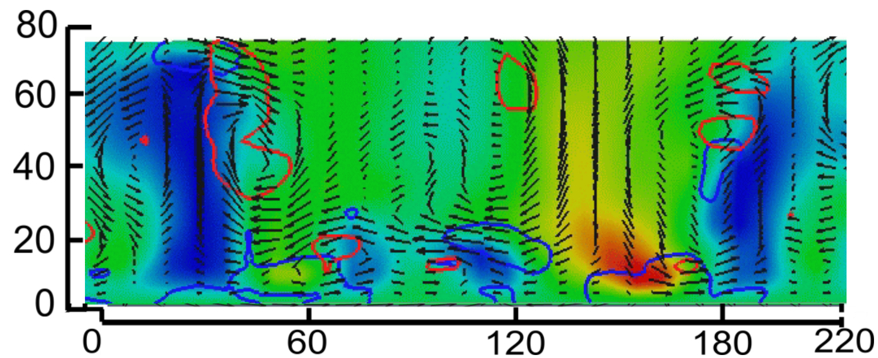
$$C_f = \frac{12}{Re_b} + 12\int_0^1 2(1-y)(\overline{-u'v'})dy$$

Viscous term and Turbulent term

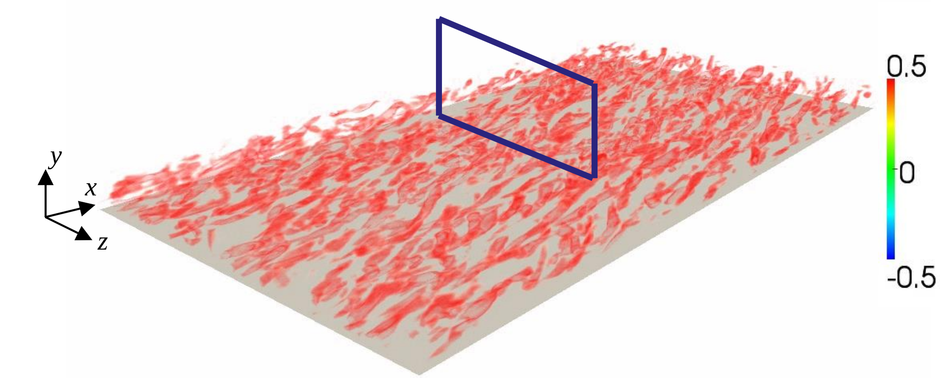
Turbulent structure – Vortex structure

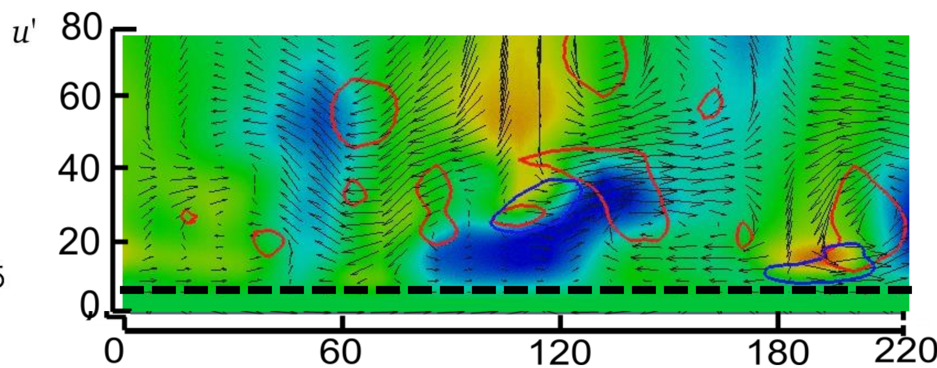
Flat





3%D





Red lines : Iso-contour of $Q^+ = 0.2$

Blue lines : Iso-contour of $\varepsilon^+ = 0.3$

Summary

- The direct numerical simulation of turbulent channel flow by standard incompressible solver *channelFoam.*
- OpenFoam is valid for analysis of turbulent flow, particularly in coupling other numerical model.

BUT

- There are few post utilities for turbulent channel flow analysis.

Future works

Other constitutive equation for porosity model

Biot's consolidation theory

$$\nabla \sigma - \nabla p + \frac{\partial d\boldsymbol{U}}{\partial t} = 0$$

$$\frac{\varphi}{K'}\frac{\partial p}{\partial t} - \frac{K}{\gamma_w}\nabla^2 p + \frac{\partial}{\partial t}(\nabla \cdot d\boldsymbol{U}) = 0$$

σ :Total stress

p :Pore pressure

$d\boldsymbol{U}$:Displacement velocity

γ_w :Specific weight of water

K' :Bulk modulus

K :Permeability

φ :Porosity

Acknowledgement

- This research work was financially supposed by a Grant-in-Aid for Young Scientists (B) (No. T247606750) from the Ministry of Education, Culture, Sports, Science and Technology of Japan and supported partly by Collaborative Research Program for Young Scientists of ACCMS and IIMC, Kyoto University.