

Turbulence and Multiphase Modeling in Computational Fluid Dynamics

Presented by: Zhaoyuan Wang

(Oct. 9, 2026)

IIHR—Hydroscience & Engineering
C. Maxwell Stanley Hydraulics Laboratory, The University of Iowa

58:160 Intermediate Mechanics of Fluids

<https://stern.lab.uiowa.edu/course-materials/me51600001-intermediate-mechanics-fluids>



Turbulence Modeling

Zhaoyuan Wang, Maysam Mousaviraad, Jun Shao,
Shanti Bhushan, Tao Xing and Fred Stern,

Interface Tracking

Zhaoyuan Wang and Fred Stern,



Outline

1. Characteristics of turbulence
2. Approaches to predicting turbulent flows
3. Reynolds averaging
4. RANS equations and unknowns
5. The Reynolds-Stress Equations
6. The closure problem of turbulence
7. Characteristics of wall-bound turbulent flows
8. Turbulence models and ranges of applicability
 - 7.1. RANS
 - 7.2. LES/DES
 - 7.3. DNS
9. Example: diffuser



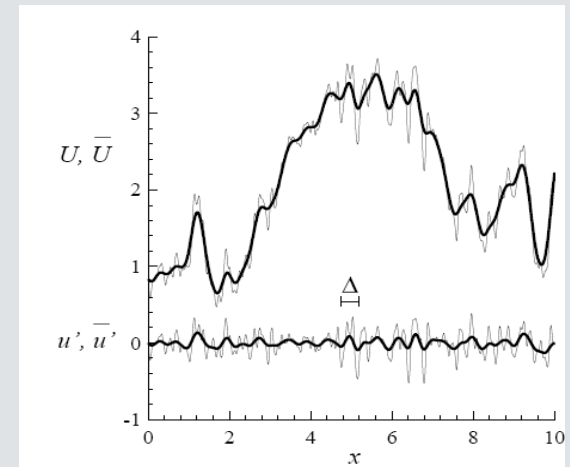
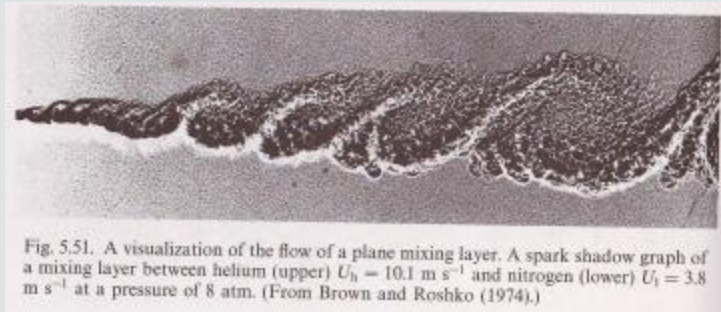
Laminar flow profile
(Reynold's number $\leq \sim 2300$)



Turbulent flow profile
(Reynold's number $\geq \sim 3000$)

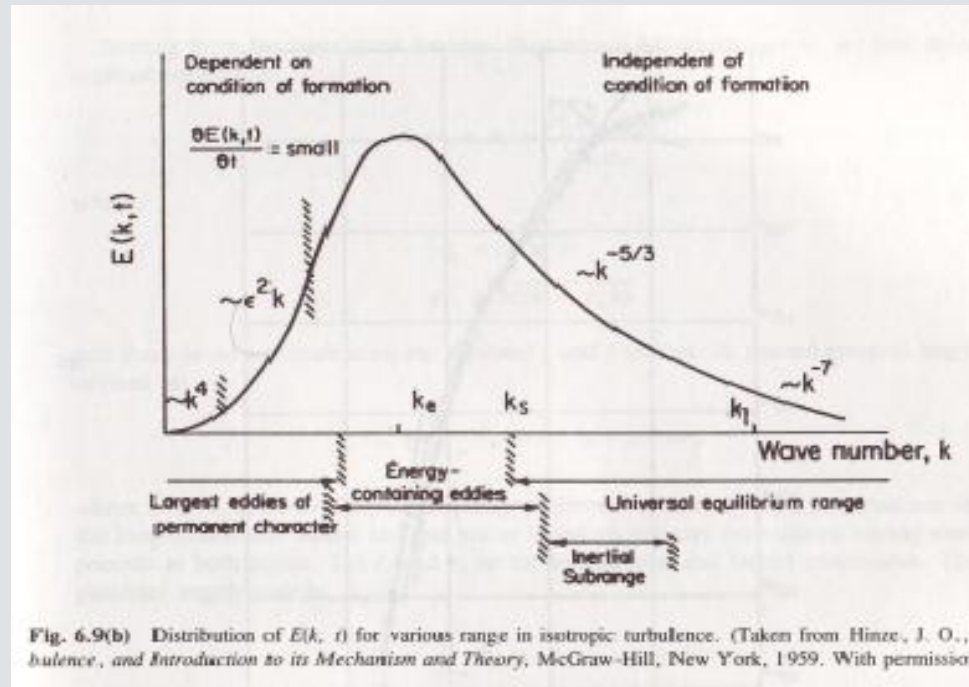
Characteristics of turbulence

- **Randomness and fluctuation:** $u = U + u'$



- **Nonlinearity:** Reynolds stresses from the nonlinear convective terms
- **Diffusion:** enhanced diffusion of momentum, energy etc.
- **Vorticity/eddies/energy cascade:** vortex stretching

Characteristics of turbulence

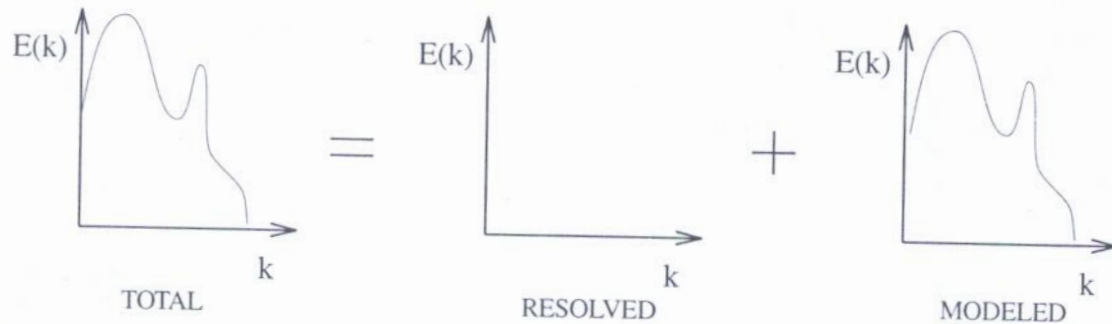


- **Dissipation**: occurs at smallest scales
- **Three-dimensional**: fluctuations are always 3D
- **Coherent structures**: responsible for a large part of the mixing
- **A broad range of length and time scales**: making DNS very difficult

Approaches to predicting turbulent flows

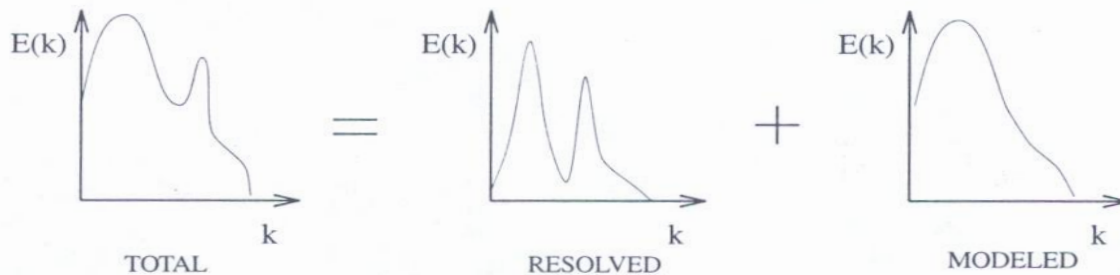
- AFD, EFD and CFD:
 - **AFD**: No analytical solutions exist
 - **EFD**: Expensive, time-consuming, and sometimes impossible (e.g. fluctuating pressure within a flow)
 - **CFD**: Promising, the need for turbulence modeling
- Another classification scheme for the approaches
 - **The use of correlations**: $C_D = f(\text{Re}())$
 - **Integral equations**: reduce PDE to ODE for simple cases
 - **One-point closure**: RANS equations + turbulent models
 - **Two-point closure**: rarely used, FFT of Two-point equations
 - **LES**: solve for large eddies while model small eddies
 - **DNS**: solve NS equations directly without any model

Deeper insights on RANS/URANS/LES



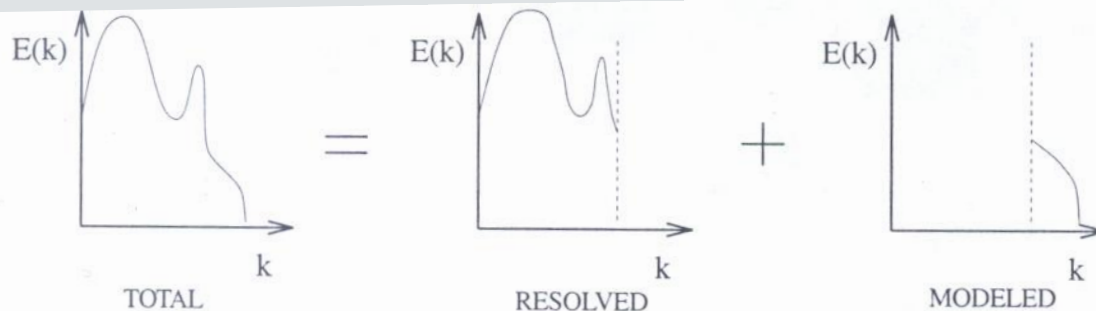
RANS

Fig. 1.1. Decomposition of the energy spectrum of the solution associated with the Reynolds Averaged Numerical Simulation (symbolic representation).



URANS

Fig. 1.2. Decomposition of the energy spectrum of the solution associated with the Unsteady Reynolds Averaged Numerical Simulation approach, when a predominant frequency exists (symbolic representation).



LES

Fig. 1.3. Decomposition of the energy spectrum in the solution associated with large-eddy simulation (symbolic representation).

Reynolds Averaging

- **Time averaging**: for stationary turbulence
- **Spatial averaging**: for homogenous turbulence
- **Ensemble averaging**: for any turbulence
- **Phase averaging**: for turbulence with periodic motion

RANS equations and unknowns

- RANS equation

$$\rho \frac{\partial U_i}{\partial t} + \rho U_j \frac{\partial U_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} (2\mu S_{ji} - \overline{\rho u'_j u'_i})$$

- RANS equation in conservative form

$$\rho \frac{\partial U_i}{\partial t} + \rho \frac{\partial}{\partial x_j} (U_j U_i + \overline{u'_j u'_i}) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} (2\mu S_{ji})$$

- Numbers of unknowns and equations

- Unknowns: **10** = P (**1**) + U (**3**) + $(-\overline{u'_i u'_j})$ (**6**)

- Equations: **4** = Continuity (**1**) + Momentum (**3**)

The Reynolds-Stress Equation

- Derivation: Taking moments of the NS equation.

Multiply the NS equation by a fluctuating property and time average the product. Using this procedure, one can derive a differential equation for the Reynolds-stress tensor.

$$\begin{aligned} \frac{\partial \tau_{ij}}{\partial t} + U_k \frac{\partial \tau_{ij}}{\partial x_k} = & -\tau_{ik} \frac{\partial U_j}{\partial x_k} - \tau_{jk} \frac{\partial U_i}{\partial x_k} + 2\nu \overline{\frac{\partial u'_i}{\partial x_k} \frac{\partial u'_j}{\partial x_k}} \\ & + \left(\overline{\frac{u'_i}{\rho} \frac{\partial p'}{\partial x_j}} + \overline{\frac{u'_j}{\rho} \frac{\partial p'}{\partial x_i}} \right) + \frac{\partial}{\partial x_k} \left(\nu \frac{\partial \tau_{ij}}{\partial x_k} + \overline{u'_i u'_j u'_k} \right) \end{aligned}$$

- NEW Equations: **6** = 6 equation for the Reynolds stress tensor
- NEW Unknowns: **22** = 6 + 6 + 10

$$\left(\overline{\frac{u'_i}{\rho} \frac{\partial p'}{\partial x_j}} + \overline{\frac{u'_j}{\rho} \frac{\partial p'}{\partial x_i}} \right) \rightarrow \mathbf{6 \text{ unknowns}}$$

$$2\nu \overline{\frac{\partial u'_i}{\partial x_k} \frac{\partial u'_j}{\partial x_k}} \rightarrow \mathbf{6 \text{ unknowns}}$$

$$\overline{u'_i u'_j u'_k} \rightarrow \mathbf{10 \text{ unknowns}}$$

The closure problem of turbulence

- Because of the non-linearity of the Navier-Stokes equation, as we take higher and higher moments, we generate additional unknowns at each level.
- In essence, **Reynolds averaging** is a brutal simplification that **loses much of the information** contained in the Navier-Stokes equation.
- The function of **turbulence modeling** is to devise **approximations for the unknown correlations** in terms of flow properties that are known so that a sufficient number of equations exist.
- In making such approximations, we close the system.

Characteristics of Wall-Bound Turbulent Flows

- The turbulent boundary layer (zero-pressure gradient) has universal velocity distribution near the wall (inner-layer) (Clauser 1951)

$$\frac{u}{u_\tau} = u^+ = \frac{yu_\tau}{\nu} = y^+ \quad : y^+ < 5$$

$$u^+ = \frac{1}{0.4} \ln(y^+) + 5.1 \quad : 10^4 > y^+ \geq 30$$

$$\frac{k}{u_\tau^2} = 0.3 \quad : 10^4 > y^+ \geq 30$$

- The velocity defect when plotted vs. y/δ collapse on a single curve (outer-layer)

$$\frac{u_\infty - u}{u_\tau} = 9.6 \left(1 - \frac{y}{\delta}\right)^2$$

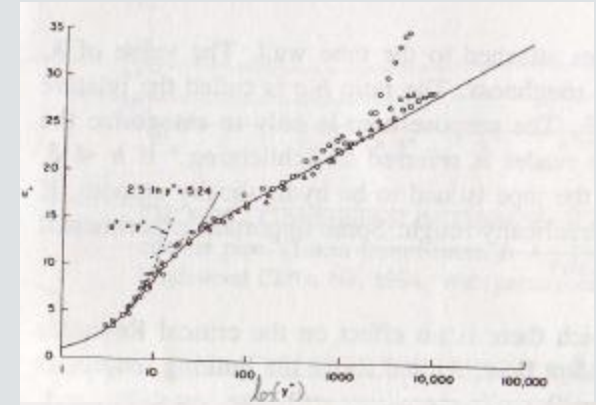
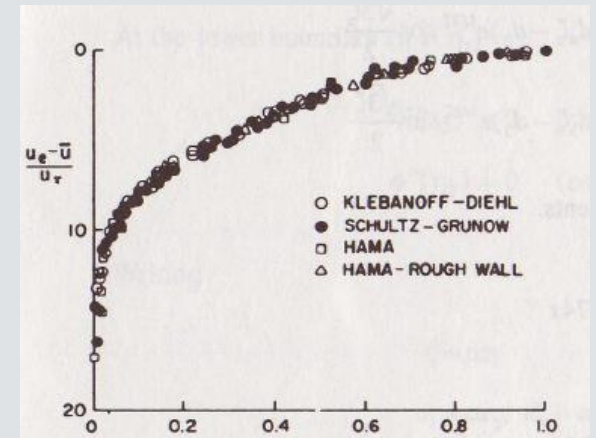


Fig. 6.11 Mean velocity distribution near a smooth wall. (Taken from Cebeci, T. and Smith, A. M. O., *Analysis of Turbulent Boundary Layers*, Academic Press, New York, 1974. With permission.)



Turbulent models (RANS)

- Boussinesq eddy-viscosity approximation
- Algebraic (zero-equation) models
 - Mixing length
 - Cebeci-Smith Model
 - Baldwin-Lomax Model
- One-equation models
 - Baldwin-Barth model
 - Spalart-Allmaras model
- Two-equation models
 - k - ε model
 - k - ω model
- Four-equation (v2f) models
- Reynolds-stress (seven-equation) models

Turbulent models (RANS)

- Boussinesq eddy-viscosity approximation

$$-\overline{u'_i u'_j} = \nu_T \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} k$$

- Dimensional analysis shows: $\nu_T = C_\mu q L$, where q is a turbulence velocity scale and L is a turbulence length scale. Usually $q = \sqrt{2k}$ where $k = \frac{1}{2} \overline{u'_i u'_i}$ is the turbulent kinetic energy. Models that do not provide a length scale are called **incomplete**.

Turbulent models (0-eqn RANS)

- Mixing length model: $v_T = l_{mix}^2 \left| \frac{dU}{dy} \right|$
- Assume $l_{mix} = \alpha \delta(x)$ for free shear flow, then
 - $\alpha=0.180$ for far wake
 - $\alpha=0.071$ for mixing layer
 - $\alpha=0.098$ for plane jet
 - $\alpha=0.080$ for round jet
- Comments:
 - Reliable only for free shear flows with different α values
 - Not applicable to wall-bounded flows

Turbulent models (0-eqn RANS)

Cebeci-Smith Model (Two-layer model)

$$v_T = \begin{cases} v_{Ti}, & y \leq y_m \\ v_{To}, & y > y_m \end{cases} \quad \text{Where } y_m \text{ is the smallest value of } y \text{ for } v_{Ti} = v_{To}$$

$$\text{Inner layer: } v_{Ti} = l_{mix}^2 \left[\left(\frac{\partial U}{\partial y} \right)^2 + \left(\frac{\partial V}{\partial x} \right)^2 \right]^{\frac{1}{2}} \quad l_{mix} = \kappa y \left[1 - e^{-\frac{y^+}{A_0^+}} \right]$$

$$\text{Outer layer: } v_{To} = \alpha U_e \delta_v^* F_{Kleb}(y; \delta)$$

$$\text{Closure coefficients: } \kappa = 0.40 \quad \alpha = 0.0168 \quad A^+ = 26 \left[1 + y \frac{dP}{dx} \frac{1}{\rho U_t^2} \right]^{\frac{-1}{2}}$$

$$F_{Kleb}(y; \delta) = \left[1 + 5.5 \left(\frac{y}{\delta} \right)^6 \right]^{-1} \quad \delta_v^* = \int_0^\delta \left(1 - \frac{U}{U_e} \right) dy$$

Comments:

Three key modifications to mixing length model;

Applicable to wall-bounded flows;

Applicable to 2D flows only;

Not reliable for separated flows;

δ, δ_v^*, U_e difficult to determine in some cases;

Turbulent models (0-eqn RANS)

Baldwin-Lomax Model (Two-layer model)

$$v_T = \begin{cases} v_{Ti}, & y \leq y_m \\ v_{To}, & y > y_m \end{cases} \quad \text{Where } y_m \text{ is the smallest value of } y \text{ for } v_{Ti} = v_{To}$$

Inner layer: $v_{Ti} = l_{mix}^2 |\omega|$ $l_{mix} = \kappa y \left[1 - e^{-\frac{y^+}{A_0^+}} \right]$

Outer layer: $v_{To} = \alpha C_{cp} F_{wake} F_{Kleb} \left(y; \frac{y_{max}}{C_{Kleb}} 0 \right)$

Closure coefficients: $\kappa = 0.40$ $\alpha = 0.0168$ $A^+ = 26$ $C_{cp} = 1.6$ $C_{Kleb} = 0.3$ $C_{wk} = 1$

$$F_{Kleb} \left(y; \frac{y_{max}}{C_{Kleb}} 0 \left[1 + 5.5 \left(\frac{y}{\frac{y_{max}}{C_{Kleb}}} 0^6 \right) \square^{-1} \right] \right) \quad \omega = \left[\left(\frac{\partial V}{\partial x} - \frac{\partial U}{\partial y} \right)^2 + \left(\frac{\partial W}{\partial y} - \frac{\partial V}{\partial z} \right)^2 + \left(\frac{\partial U}{\partial z} - \frac{\partial W}{\partial x} \right)^2 \right]^{\frac{1}{2}}$$

Comments:

Applicable to 3D flows;

Not reliable for separated flows;

No need to determine δ, δ_v^*, U_e

Turbulent models (1-eqn RANS)

Baldwin-Barth model

Kinematic eddy viscosity : $\nu_T = C_\mu \nu \tilde{R}_T D_1 D_2$

Turbulence Reynolds number :

$$\frac{\partial(\nu \tilde{R}_T)}{\partial t} + U_j \frac{\partial(\nu \tilde{R}_T)}{\partial x_j} = (C_{\varepsilon 2} f_2 - C_{\varepsilon 1}) \sqrt{\nu \tilde{R}_T P} + \left(\nu + \frac{\nu_T}{\sigma_\varepsilon} \right) \frac{\partial^2(\nu \tilde{R}_T)}{\partial x_k \partial x_k} - \frac{1}{\sigma_\varepsilon} \frac{\partial \nu_T}{\partial x_k} \frac{\partial(\nu \tilde{R}_T)}{\partial x_k}$$

Closure coefficients and auxiliary relations:

$$C_{\varepsilon 1} = 1.2 \quad C_{\varepsilon 2} = 2.0 \quad C_\mu = 0.09 \quad A_0^+ = 26 \quad A_2^+ = 10 \quad \frac{1}{\sigma_\varepsilon} = (C_{\varepsilon 1} - C_{\varepsilon 2}) \frac{\sqrt{C_\mu}}{\kappa^2} \quad \kappa = 0.41$$

$$P = \nu_T \left[\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \frac{\partial U_k}{\partial x_k} \right] \quad D_1 = 1 - e^{-\frac{y^+}{A_0^+}} \quad D_2 = 1 - e^{-\frac{y^+}{A_2^+}}$$

$$f_2 = \frac{C_{\varepsilon 1}}{C_{\varepsilon 2}} + \left(1 - \frac{C_{\varepsilon 1}}{C_{\varepsilon 2}} \right) \left(\frac{1}{\kappa y^+} + D_1 D_2 \right) \cdot \left[\sqrt{D_1 D_2} + \frac{1}{\sqrt{D_1 D_2}} \left(\frac{D_2}{A_0^+} e^{-\frac{y^+}{A_0^+}} + \frac{D_1}{A_2^+} e^{-\frac{y^+}{A_2^+}} \right) \right]$$

Turbulent models (1-eqn RANS)

Spalart-Allmaras model

Kinematic eddy viscosity : $\nu_T = \tilde{\nu} f_{v1}$

Eddy viscosity equation:

$$\frac{\partial \tilde{\nu}}{\partial t} + U_j \frac{\partial \tilde{\nu}}{\partial x_j} = c_{b1} \tilde{S} \tilde{\nu} - c_{w1} f_w \left(\frac{\tilde{\nu}}{d} \right)^2 + \frac{1}{\sigma} \frac{\partial}{\partial x_k} \left[(\nu + \tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial x_k} \right] + \frac{c_{b1}}{\sigma} \frac{\partial \tilde{\nu}}{\partial x_k} \frac{\partial \tilde{\nu}}{\partial x_k}$$

Closure coefficients and auxiliary relations:

$$\begin{aligned} c_{b1} &= 0.1355 & c_{b2} &= 0.622 & c_{v1} &= 7.1 & \sigma &= \frac{2}{3} & c_{w1} &= \frac{c_{b1}}{\kappa^2} + \frac{(1 + c_{b2})}{\sigma} \\ c_{w2} &= 0.3 & c_{w3} &= 2.0 & \kappa &= 0.41 & \chi &= \frac{\tilde{\nu}}{\nu} & g &= r + c_{w2}(r^6 - r) & r &= \frac{\tilde{\nu}}{\tilde{S} \kappa^2 d^2} \\ f_{v1} &= \frac{\chi^3}{\chi^3 + c_{v1}^3} & f_{v2} &= 1 - \frac{\chi}{1 + \chi f_{v1}} & f_w &= g \left[\frac{1 + c_{w3}^6}{g^6 + c_{w3}^6} \right]^{\frac{1}{6}} & \tilde{S} &= S + \frac{\tilde{\nu}}{\kappa^2 d^2} f_{v2} & S &= \sqrt{2 \Omega_{ij} \Omega_{ij}} \end{aligned}$$

Turbulent models (1-eqn RANS)

Comments on one-equation models:

1. One-equation models based on turbulence kinetic energy are **incomplete** as they relate the turbulence length scales to some typical flow dimension. They are rarely used.
2. One-equation models based on an equation for the eddy viscosity are **complete** such as Baldwin-Barth model and Spalart-Allmaras model.
3. They circumvent the need to specify a dissipation length by expressing the decay, or dissipation, of the eddy viscosity in terms of spatial gradients.
- 4 **Spalart-Allmaras** model can predicts better results than Baldwin-Barth model, and much better results for separated flow than Baldwin-Barth model and algebraic models.
- 5 Also most of DES simulations are based on the Spalart-Allmaras model.

Turbulent models (2-eqn RANS)

k-ε model:

$$\nu_T = C_\mu \frac{k^2}{\varepsilon}$$

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]$$

$$\frac{\partial \varepsilon}{\partial t} + U_j \frac{\partial \varepsilon}{\partial x_j} = C_{\varepsilon 1} \frac{\varepsilon}{k} \tau_{ij} \frac{\partial U_i}{\partial x_j} - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_T}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right]$$

K: turbulent kinetic energy

ε: rate of dissipation

ω: specific rate of dissipation

$$C_{\varepsilon 1} = 1.44$$

$$C_{\varepsilon 2} = 1.92$$

$$C_\mu = 0.09$$

$$\sigma_k = 1.0$$

$$\sigma_\varepsilon = 1.3$$

$$\omega = \frac{\varepsilon}{(C_\mu k)}$$

$$l = C_\mu \frac{k^{\frac{3}{2}}}{\varepsilon}$$

k-ω model:

$$\nu_T = \frac{k}{\omega}$$

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[(\nu + \sigma^* \nu_T) \frac{\partial k}{\partial x_j} \right]$$

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha \frac{\omega}{k} \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[(\nu + \sigma \nu_T) \frac{\partial \omega}{\partial x_j} \right]$$

$$\alpha = \frac{13}{25}$$

$$\beta = \beta_0 f_\beta$$

$$\beta^* = \beta_0^* f_{\beta^*}$$

$$\sigma = \frac{1}{2}$$

$$\sigma^* = \frac{1}{2}$$

$$\beta_0 = \frac{9}{125}$$

$$f_\beta = \frac{1 + 70\chi_\omega}{1 + 80\chi_\omega}$$

$$\chi_\omega = \left| \frac{\Omega_{ij} \Omega_{jk} S_{ki}}{(\beta_0^* \omega)^3} \right| \quad \beta_0^* = \frac{9}{100} \quad f_{\beta^*} = \begin{cases} 1, & \chi_k \leq 0 \\ \frac{1 + 680\chi_k^2}{1 + 400\chi_k^2} & \chi_k > 0 \end{cases}$$

$$\chi_k = \frac{1}{\omega^3} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}$$

$$\varepsilon = \beta^* \omega k$$

$$l = \frac{k^{\frac{1}{2}}}{\omega}$$

Turbulent models (2-eqn RANS)

Comments on two-equation models:

1. Two-equation models are complete;
2. k - ϵ and k - ω models are the most widely used two-equation models and a lot of versions exist. For example, a popular variant of k - ω model introduced by Menter has been used in our research code CFDSHIP-IOWA. There are also a lot of low-Reynolds-number versions with different damping functions. k - ω model inner region and k - ϵ in the outer region.
3. k - ω model shows better results than k - ϵ model for flows with adverse pressure gradient and separated flows as well as better numerical stability.

Turbulent models (4-eqn RANS)

v2f-k ε model:

$$\nu_T = C_\mu \overline{v^2} T$$

V^2 , the velocity fluctuation normal to the streamlines; f elliptic relaxation function

$$\frac{D\overline{v^2}}{Dt} = kf - \varepsilon \frac{\overline{v^2}}{k} + \frac{\partial}{\partial x_j} \left[(\nu + \nu_t) \frac{\partial \overline{v^2}}{\partial x_j} \right]$$

$$L^2 \nabla^2 f - f = \frac{C_1}{T} \left(\frac{\overline{v^2}}{k} - \frac{2}{3} \right) - C_2 \frac{P_k}{k}$$

$$\frac{Dk}{Dt} = P - \varepsilon + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]$$

$$\frac{D\varepsilon}{Dt} = C_{\varepsilon 1} \frac{\varepsilon}{k} P - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right]$$

$$C_1 = 0.4 \quad C_2 = 0.3 \quad C_L = 0.3 \quad C_\eta = 70 \quad C_{\varepsilon 2} = 1.9 \quad \sigma_\varepsilon = 1.0$$

$$C_{\varepsilon 1} = 1.3 + \frac{0.25}{\left[1 + \left(\frac{C_\mu d}{2L} \right)^2 \right]^4}$$

$$T = \max \left\{ \frac{k}{\varepsilon}, 6 \sqrt{\frac{\nu}{\varepsilon}} \right\}$$

$$L = \max \left\{ C_L \frac{k^{\frac{3}{2}}}{\varepsilon}, C_\eta \left(\frac{\nu^3}{\varepsilon} \right)^{\frac{1}{4}} \right\}$$

v2f-k ω model:

$$\nu_T = C_\mu \frac{k^n \overline{v^2}^{1-n}}{\omega}$$

$$\frac{D\overline{v^2}}{Dt} = kf - 6\overline{v^2} \frac{\varepsilon}{k} + \frac{\partial}{\partial x_j} \left[(\nu + \nu_t) \frac{\partial \overline{v^2}}{\partial x_j} \right]$$

$$L^2 \nabla^2 - f = \frac{1}{T} (C_1 - 1) \left(\frac{\overline{v^2}}{k} - \frac{2}{3} \right) - 5 \frac{1}{T} \frac{\overline{v'^2}}{k} - C_2 \frac{P_k}{k}$$

$$\frac{Dk}{Dt} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta^* \omega k + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]$$

$$\frac{D\omega}{Dt} = \alpha \frac{\omega}{k} \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta \omega^2 \left(\frac{\overline{v^2}}{k} \right)^{1-n} + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right]$$

$$\alpha = 0.45977$$

$$\beta^* = 0.09$$

$$\beta = \frac{3}{40}$$

$$\sigma_k = 1.0$$

$$\sigma_\omega = 1.5$$

$$T = \max \left\{ \frac{k}{\varepsilon}, 6 \sqrt{\frac{\nu}{\varepsilon}} \right\}$$

$$L = \max \left\{ C_L \frac{k^{\frac{3}{2}}}{\varepsilon}, C_\eta \left(\frac{\nu^3}{\varepsilon} \right)^{\frac{1}{4}} \right\}$$

Turbulent models (4-eqn RANS)

Comments on four-equation models:

1. For two-equation models a major problem is that it is hard to specify the proper conditions to be applied near walls.
2. Durbin suggested that the problem is that the Reynolds number is low near a wall and that the impermeability condition (zero normal velocity) is far more important. That is the motivation for the equation for the normal velocity fluctuation.
3. It was found that the model also required a damping function f , hence the name v_2f model.
4. They appear to give improved results at essentially the same cost as the $k-\varepsilon$ and $k-\omega$ models especially for separated flows. Hopefully the $v_2f-k\omega$ model can have better numerical stability than $v_2f-k\varepsilon$ model as their counterparts behave in two-equation models.

Turbulent models (7-eqn RANS)

Some of the most noteworthy types of applications for which models based on the Boussinesq approximation fail are:

1. Flows with sudden changes in mean strain rate
2. Flow over curved surfaces
3. Flow in ducts with secondary motions
4. Flow in rotating fluids
5. Three-dimensional flows
6. Flows with boundary-layer separation

In Reynolds-stress models, the equations for the Reynolds stress tensor are modeled and solved along with the ε -equation:

$$\frac{\partial \tau_{ij}}{\partial t} + U_k \frac{\partial \tau_{ij}}{\partial x_k} = -\tau_{ik} \frac{\partial U_j}{\partial x_k} - \tau_{jk} \frac{\partial U_i}{\partial x_k} + 2\nu \frac{\partial \overline{u'_i}}{\partial x_k} \frac{\partial \overline{u'_j}}{\partial x_k} + \left(\frac{\overline{u'_i \partial p'}}{\rho \partial x_j} + \frac{\overline{u'_j \partial p'}}{\rho \partial x_i} \right) + \frac{\partial}{\partial x_k} \left(\nu \frac{\partial \tau_{ij}}{\partial x_k} + \overline{u'_i u'_j u'_k} \right)$$

Turbulent models (7-eqn RANS)

These are some versions of Reynolds stress models:

1. LRR rapid pressure-strain model
2. Lumley pressure-strain model
3. SSG pressure-strain model
4. Wilcox stress- model

Comments on Reynolds stress models:

1. Reynolds stress models require the solution of **seven additional PDEs** and those equation are even harder to solve than the two-equation models.
2. Although Reynolds stress models have greater potential to represent turbulent flow more correctly, their success so far has been moderate.
3. There is a lot of current research in this field, and new models are often proposed. Which model is best for which kind of flow is not clear due to the fact that in many attempts to answer this question numerical errors were too large to allow clear conclusions to be reached.

Turbulent models (LES)

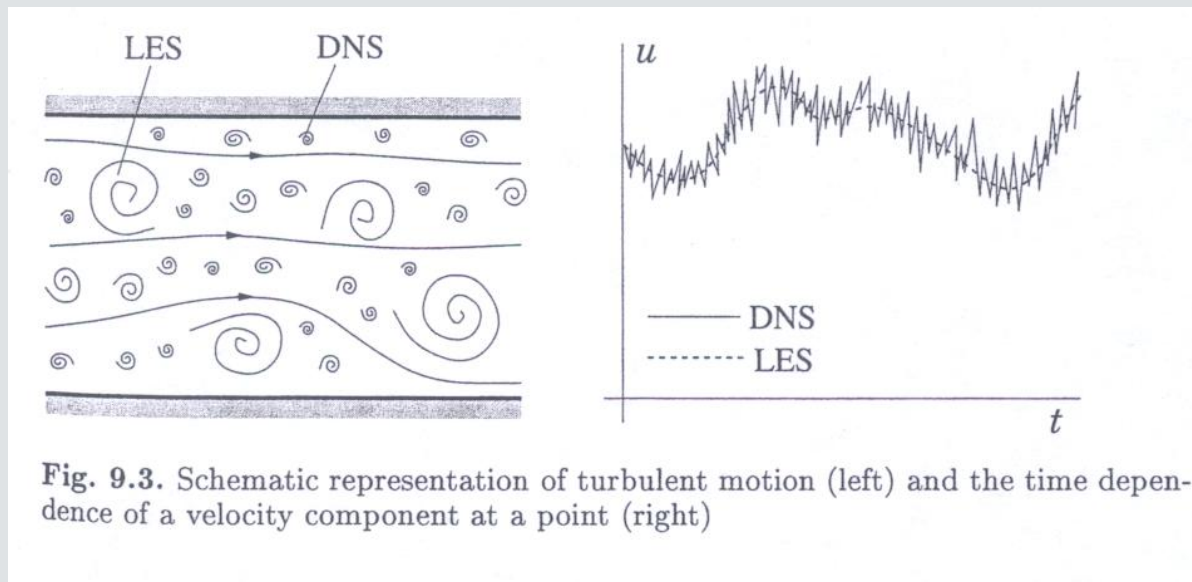


Fig. 9.3. Schematic representation of turbulent motion (left) and the time dependence of a velocity component at a point (right)

- Large scale motions are generally much more energetic than the small scale ones.
- The size and strength of large scale motions make them to be the most effective transporters of the conserved properties.
- LES treats the large eddies more exactly than the small ones may make sense
- LES is 3D, time dependent and expensive but much less costly than DNS.

Turbulent models (LES, filtering)

- LES needs a velocity field that contains only the large scale components of the total field, which is achieved by filtering the velocity field (Leonard, 1974)

$$\bar{u}_i(x) = \int G(x - \xi) u_i(\xi) d\xi$$

- $G(x-\xi)$ is the filter kernel, is a localized function, which includes a **Gaussian**, a **box filter** (a simple local average) and a **cutoff** (a filter which eliminates all Fourier coefficients belonging to wavenumbers above a cutoff)
- Each filter has a length scale associated with it, Δ .
- Eddies of size large than Δ are large eddies while those smaller than Δ are small eddies and need to be modeled.

$$G(x - \xi) = \begin{cases} \frac{1}{\bar{\Delta}} & \text{if } |x - \xi| \leq \frac{\bar{\Delta}}{2} \\ 0 & \text{otherwise} \end{cases}$$

Box or top-hat filter

$$G(x - \xi) = \left(\frac{\gamma}{\pi \bar{\Delta}^2} \right)^{1/2} \exp \left(\frac{-\gamma |x - \xi|^2}{\bar{\Delta}^2} \right)$$

Gaussian filter

$$G(x - \xi) = \frac{\sin(k_c(x - \xi))}{k_c(x - \xi)}$$
$$k_c = \frac{\pi}{\Delta}$$

Cutoff filter

Turbulent models (LES, Governing Equations)

- Filtered Navier-Stokes equations (constant density, incompressible):

$$\frac{\partial(\rho \bar{u}_i)}{\partial x_i} = 0, \quad \frac{\partial(\rho \bar{u}_i)}{\partial t} + \frac{\partial(\rho \overline{u_i u_j})}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right]$$

- Note that: $\overline{u_i u_j} \neq \bar{u}_i \bar{u}_j$ property of the filtering operator

$$\frac{\partial(\rho \bar{u}_i)}{\partial t} + \frac{\partial(\rho \overline{u_i u_j} + \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right]$$

$$\frac{\partial(\rho \bar{u}_i)}{\partial t} + \frac{\partial(\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right] + \frac{\partial(\tau_{ij}^S)}{\partial x_j}$$

Introducing Subgrid-scale Reynolds Stress

$$\tau_{ij}^S = -\rho(\overline{u_i u_j} - \bar{u}_i \bar{u}_j)$$

- The filter width $\Delta >$ grid size h
- The models used to approximate the SGS Reynolds stress are called subgrid-scale (SGS) or subfilter-scale models.

Turbulent models (LES, Smagorinsky model)

- The earliest and most commonly used subgrid scale model is one proposed by Smagorinsky (1963), which is an eddy viscosity model.
- As the increased transport and dissipation are due to the viscosity in laminar flow, it seems reasonable to assume that

$$\tau_{i,j}^S - \frac{1}{3}\tau_{kk}^S\delta_{i,j} = \mu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) = 2\mu_t \bar{S}_{ij}$$

$\bar{S}_{ij}$ Strain rate of the large scale or resolved field

μ_t Eddy viscosity $\mu_t = C_S^2 \rho \Delta^2 |\bar{S}|$

$C_S \approx 0.2$ Model constant

- Drawbacks:
 1. C_S is not constant
 2. Changes of C_S are required in all shear flows
 3. Need to be reduced near the wall.
 4. Not accurate for complex and/or higher Reynolds number flows.

Turbulent models (LES, Scale-similarity model and Dynamic model)

- **Dynamic model:**

1. filtered LES solutions can be filtered again using a filter broader than the previous filter to obtain a very large scale field.
2. An effective subgrid-scale field can be obtained by subtraction of the two fields.
3. model parameter can then be computed.

Advantages: 1. model parameter computed at every spatial grid point and every time step from the results of LES

2. Self-consistent subgrid-scale model

3. Automatically change the parameter near the wall and in shear flows

Disadvantages: backscatter (eddy viscosity < 0) may cause instability.

Turbulent models (DES)

- Massively separated flows at high Re usually involve both large and small scale vortical structures and very thin turbulent boundary layer near the wall
- RANS approaches are efficient inside the boundary layer but predict very excessive diffusion in the separated regions
- LES is accurate in the separated regions but is unaffordable for resolving thin near-wall turbulent boundary layers at industrial Reynolds numbers
- Motivation for DES: combination of LES and RANS. RANS inside attached boundary layer and LES in the separated regions

DES Formulation

- Modification to RANS models was straightforward by substituting the length scale d_w , which is the distance to the closest wall, with the new DES length scale, $\tilde{l}$ defined as:

$$\tilde{l} = \min(d_w, C_{DES}\Delta)$$

$$\Delta = \max(\delta_x, \delta_y, \delta_z)$$

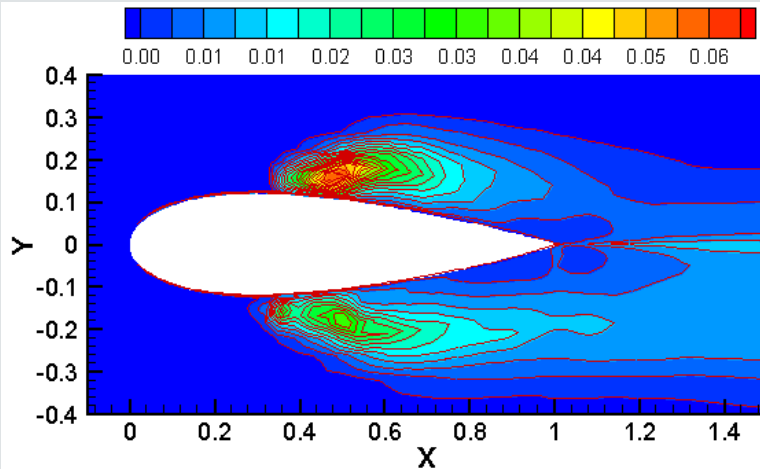
- where C_{DES} is the DES constant, Δ is the grid spacing and is based on the largest dimensions of the local grid cell, and $\delta_x, \delta_y, \delta_z$ are the grid spacing in x, y and z coordinates respectively
- Applying the above modification will result in S-A Based, standard k- ϵ (or k- ω) based and Menter's SST based DES models, etc.

$$D_{RANS}^k = \rho \beta^* k \omega = \rho k^{\frac{3}{2}} / l_{k-\omega}$$
$$D_{DES}^k = \rho \frac{k^{\frac{3}{2}}}{\tilde{l}}$$

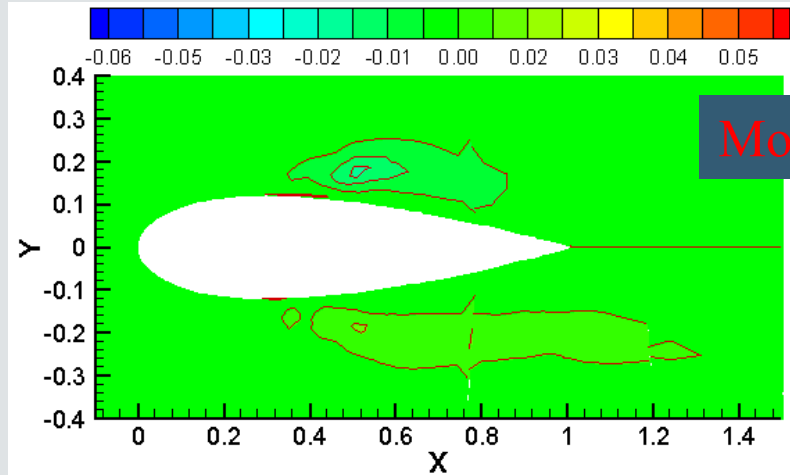
$$l_{k-\omega} = \frac{k^{\frac{1}{2}}}{(\beta^* \omega)}$$
$$\tilde{l} = \min(l_{k-\omega}, C_{DES}\Delta)$$

Resolved/Modeled/Total Reynolds stress (DES)

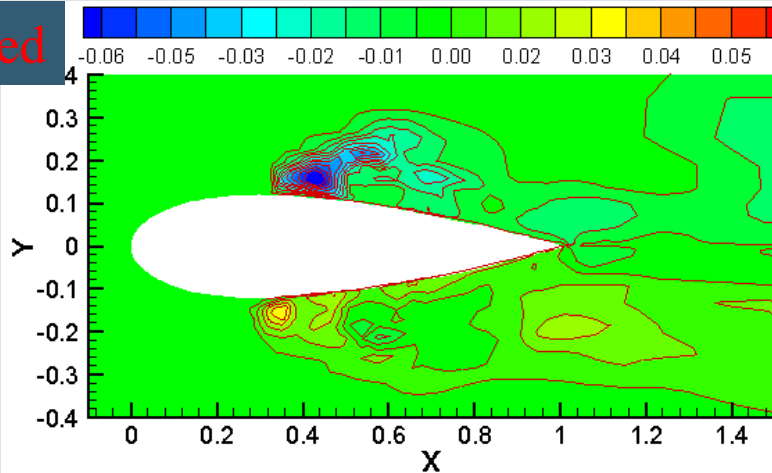
TKE



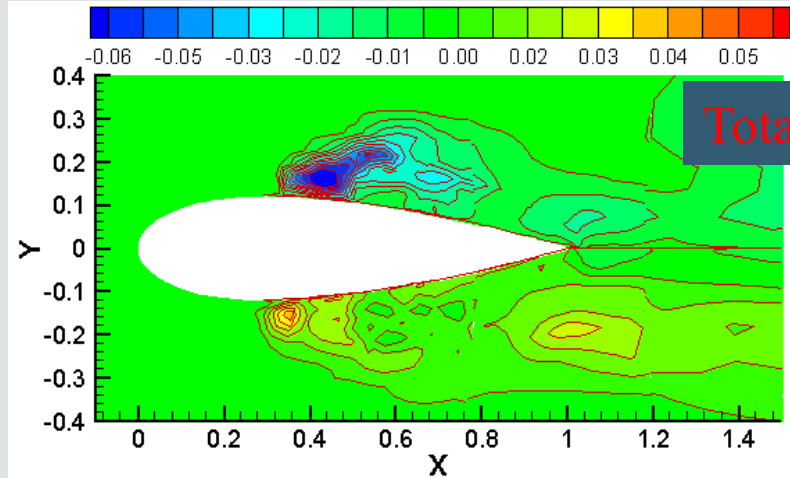
Modeled



Resolved



Total

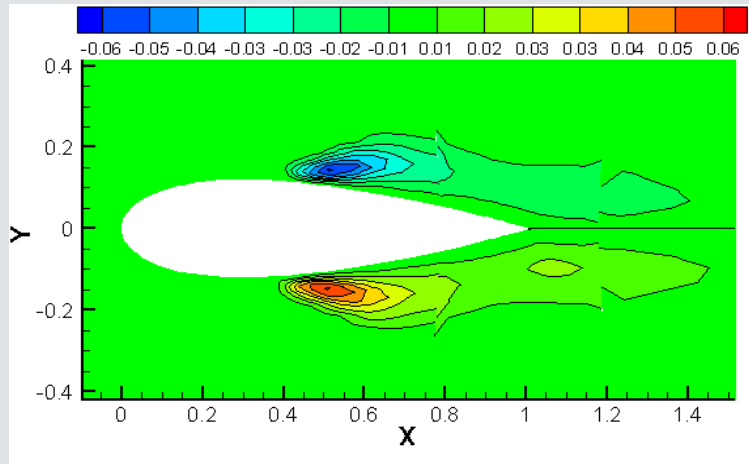


Modeled Reynolds stress:

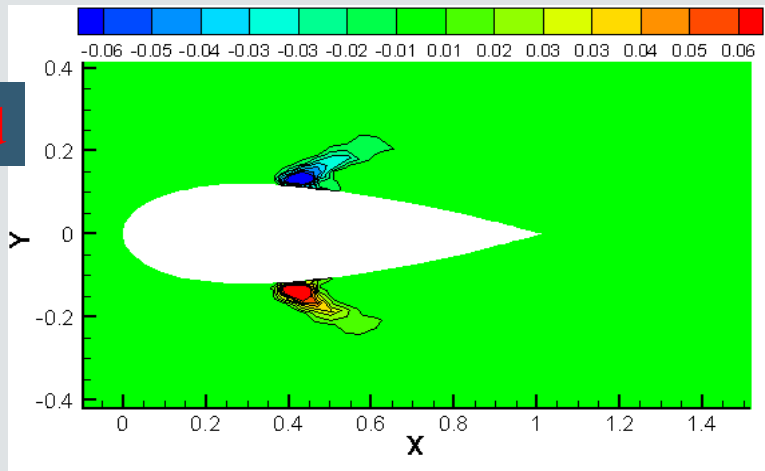
$$\overline{u'v'} = -\nu_t \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)$$

Resolved/Modeled/Total Reynolds stress (URANS)

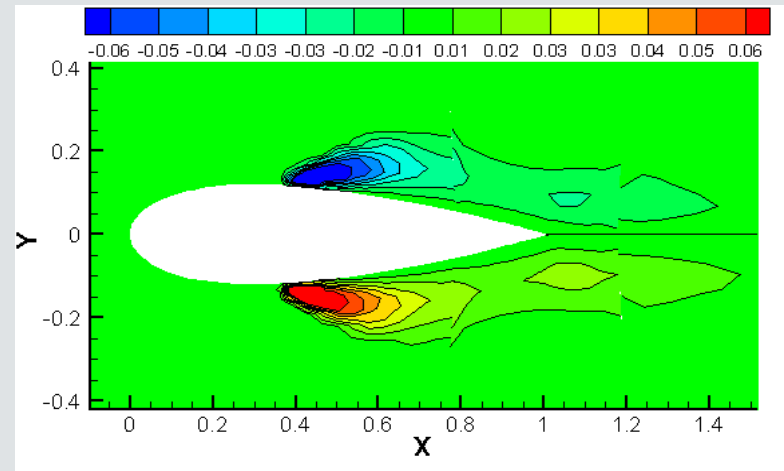
Modeled



Resolved



Total

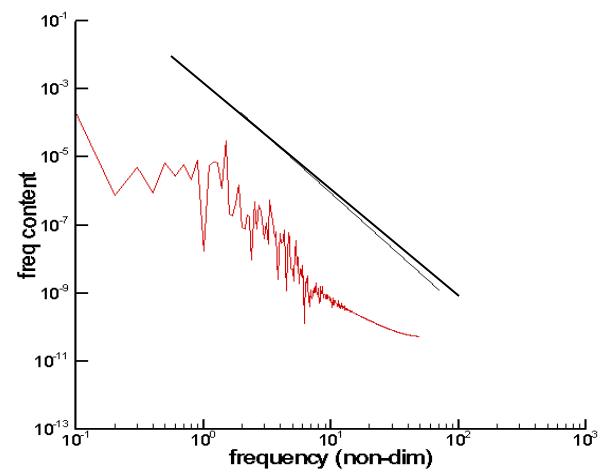
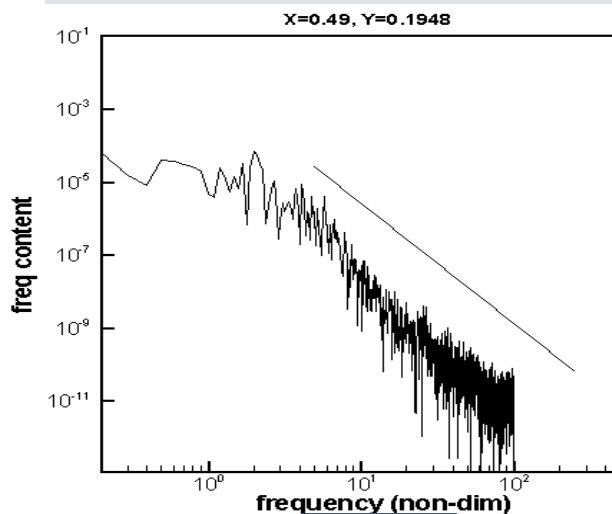
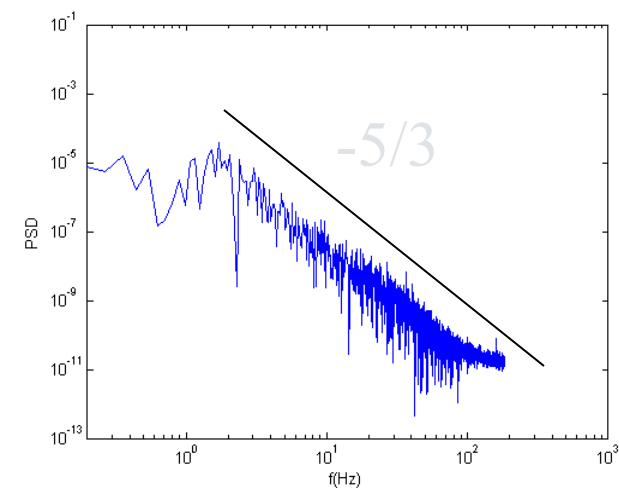
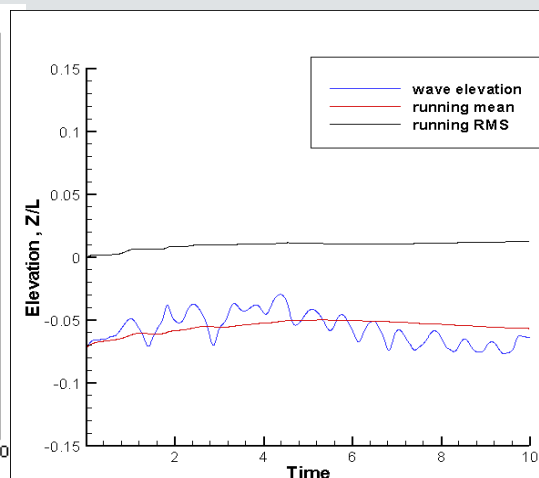
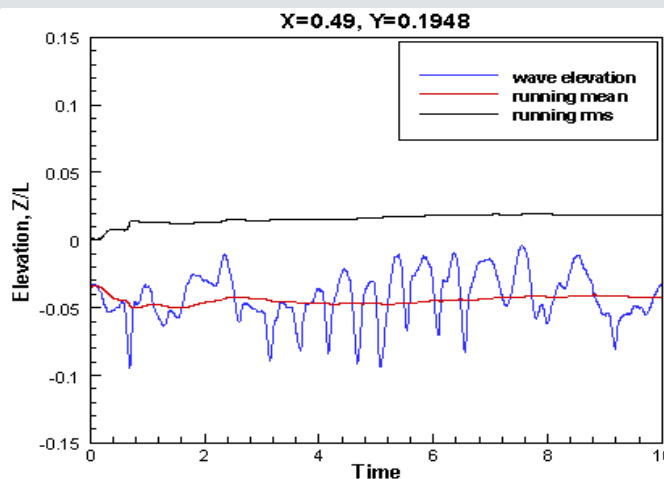
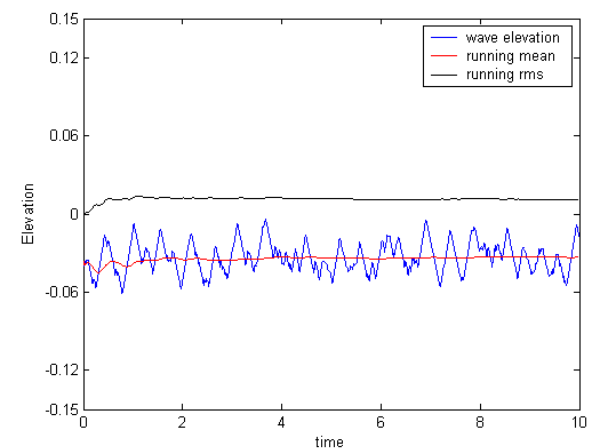


Deep insight into RANS/DES (Point oscillation on free surface with power spectral analysis)

EFD

DES

URANS



2HZ

Direct Numerical Simulation (DNS)

- **DNS is** to solve the Navier-Stokes equation directly without averaging or approximation other than numerical discretizations whose errors can be estimated (V&V) and controlled.
- **The domain** of DNS must be at least as large as the physical domain or the largest turbulent eddy (scale L)
- The **size of the grid** must be no larger than a viscously determined scale, Kolmogoroff scale, η
- The **number of grid points** in each direction must be at least L/η
- The computational cost is proportional to

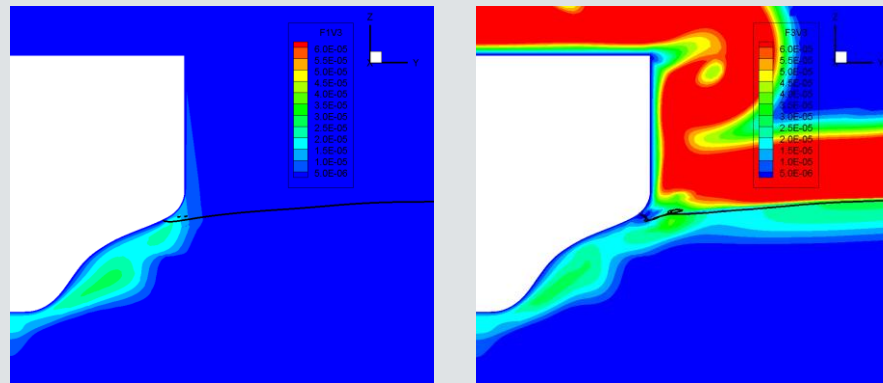
$$\text{Re}_L^{3/4} \approx (0.01\text{Re}())^{3/4}$$

- Provide detailed information on flow field
- Due to the computational cost, DNS is more likely to be a research tool, not a design tool.

Two-Phase Turbulence

- Most of the turbulence models (RANS and LES) are developed based on single-phase flows. The effect of turbulence on the interface and the interface induced turbulence are not considered.
- The eddy viscosity is found to be over-predicted in the gas area near the interface (Wang et. al, 2010).
- A buoyancy term G_b is added to the TKE

Equation (Devolder et al., 2017/2018):



Eddy viscosity for KCS wave breaking. Left: V4.5; right: V5.5.

- This additional term suppresses the turbulence level at the air-water interface

Examples (Diffuser)

- Asymmetric diffuser with separation is a good test case for turbulence models.
- An inlet channel was added at the diffuser inlet to generate fully developed velocity profile
- Boundary layer in the lower diffuser wall will separate due to the adverse pressure gradient.
- Results shown next include comparisons between **V2f** and **k- ϵ**
- **LES simulation** of this geometry can be found in:
M. Fatica, H. J. Kaltenbach, and R. Mittal, "Validation of LES in a Plain Asymmetric Diffuser", *center for turbulence research, annual research briefs, 1997*

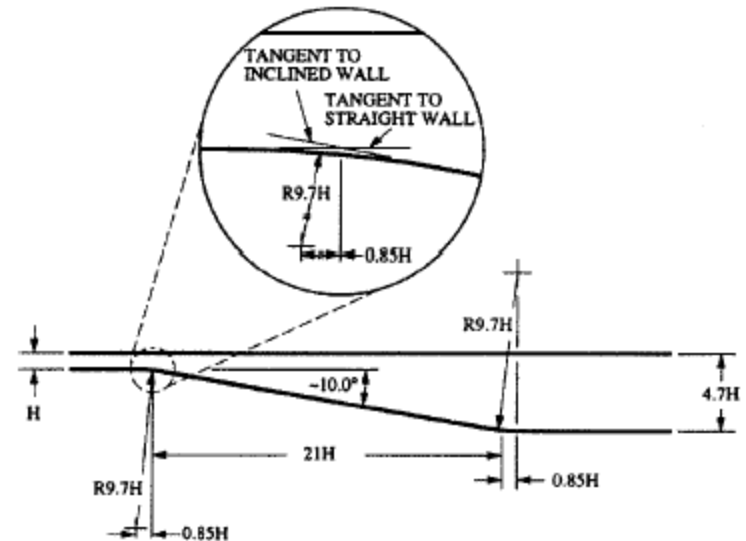


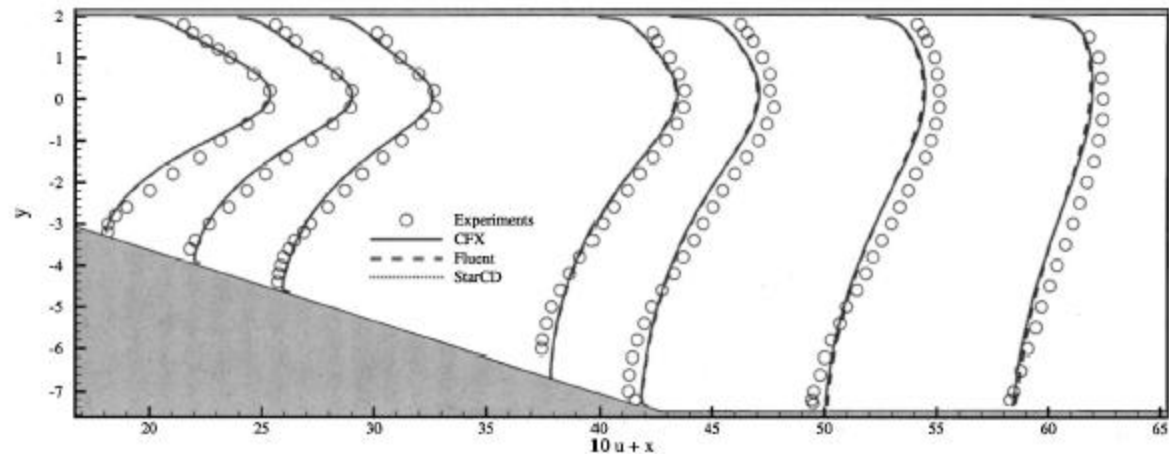
Fig. 1 Asymmetric diffuser geometry



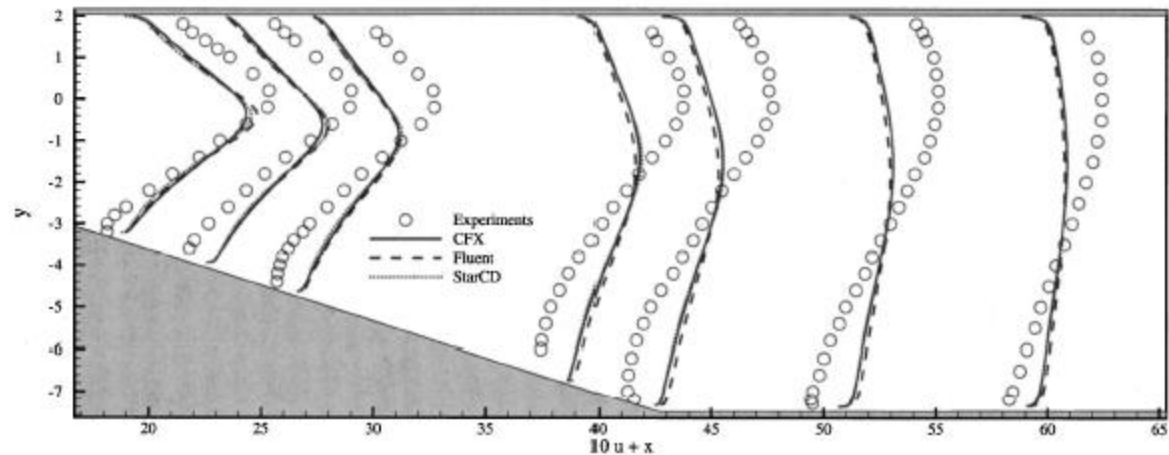
Fig. 2 Computational grid—detail of the channel-diffuser connection

Examples (Diffuser)

- Mean velocity predicted by V2f agreed very well with EFD data, particular the **separation region is captured**.
- K- ϵ model **fails to predict the separation** caused by adverse pressure gradient.

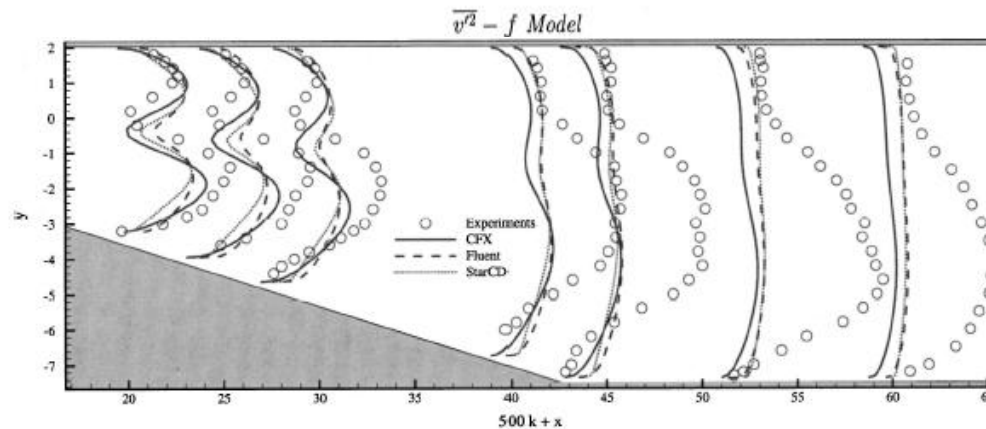
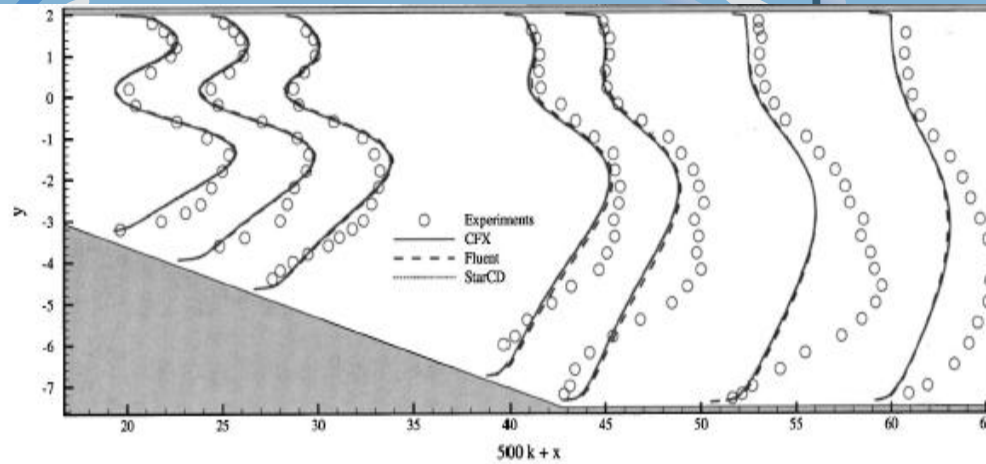


$\overline{v^2} - f$ Model



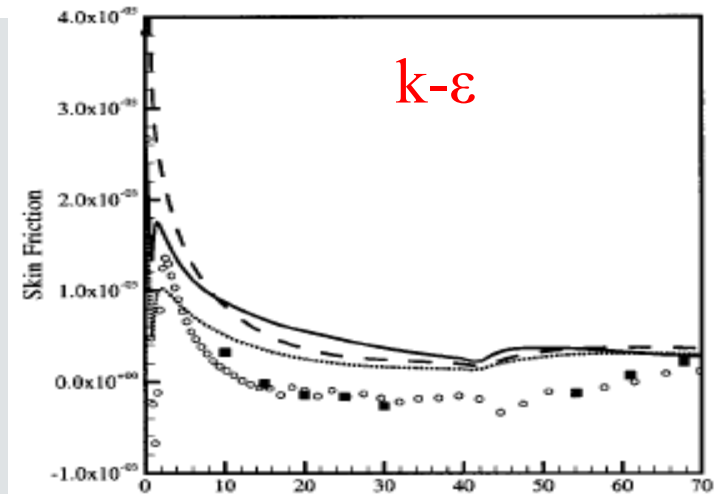
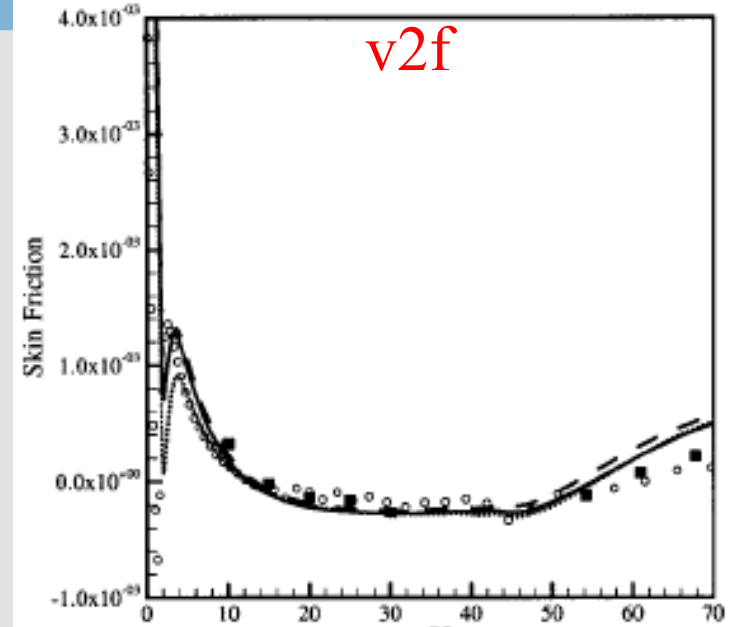
Low-Reynolds $k-\epsilon$ Model

Examples (Diffuser)



Low-Reynolds $k-\epsilon$ Model

- TKE predicted by V2f agreed better with EFD data than $k-\epsilon$ model, particular the asymmetric distribution.
- Right column is for the skin friction coefficient on the lower wall, from which the separation and reattachment point can be found.



$2x/H$

References

1. J. H. Ferziger, M. Peric, “**Computational Methods for Fluid Dynamics**,” 3rd edition, Springer, 2002.
2. D. C. Wilcox, “**Turbulence Modeling for CFD**”, 1998.
3. S.B.Pope, “Turbulent Flows,” Cambridge, 2000
4. P. A. Durbin and B. A. Pettersson Reif, “Statistical Theory and Modeling for Turbulent Flows,” John Wiley & Sons, LTD, 2001.
5. P.A. Davidson, “Turbulence: An Introduction for Scientists and Engineers,” Oxford, 2004.



Interface Tracking

Zhaoyuan Wang and Fred Stern

IIHR—Hydroscience & Engineering
C. Maxwell Stanley Hydraulics Laboratory
The University of Iowa

58:160 Intermediate Mechanics of Fluids
<https://stern.lab.uiowa.edu/course-materials/me51600001-intermediate-mechanics-fluids>
Oct. 09, 2026

Interface Tracking Methods

Interface tracking methods

- Volume of Fluid (**VOF**) Method
 - *Good mass conservation property*
 - *Low accuracy to compute curvature and normal*
- Level Set (**LS**) Method
 - *Estimation of curvature and normal more accurate*
 - *Poor mass conservation property*
- Coupled Level Set and Volume-of-Fluid (**CLSVOF**) Method
 - *Interface precisely located*
 - *Mass well conserved*

Interface modeling methods

The *level set* (LS) and *volume-of-fluid* (VOF) methods have been extensively used for the interface tracking.

- *LS method:*

- ✓ simple and easy to implement, but susceptible to the numerical dissipation and poor mass conservation;
- ✓ not adopted by popular commercial codes, mostly used in research codes: e.g., CFDSHIP-IOWA V4.5, WAVIS, SPLASH, FreSCO+, NAGISA, etc.

- *VOF method:*

- ✓ accurate and good mass conservation property, but implementation is difficult using geometric method;
- ✓ widely used in commercial, open source, and research codes: FLOW-3D, ANSYS Fluent, STAR-CCM+ , OpenFoam, Gerris (code), NFA, CFDSHIP-IOWA V6, etc.

The implementation schemes of the VOF method: *algebraic* and *geometric* methods

- *Algebraic VOF method:*

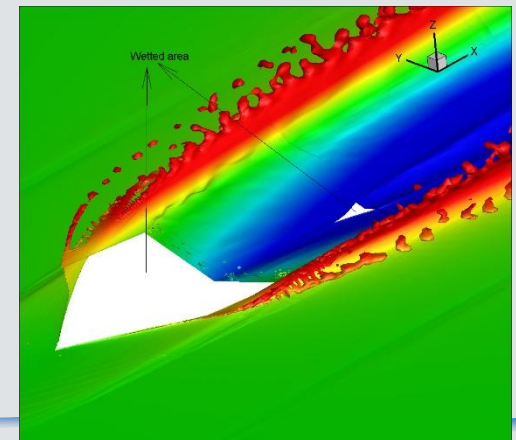
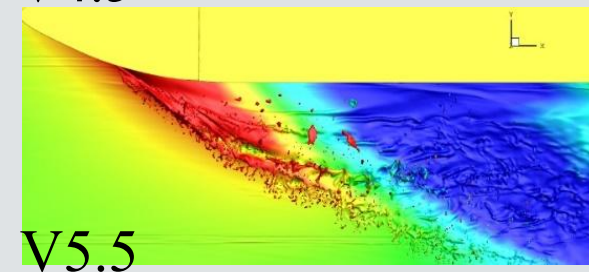
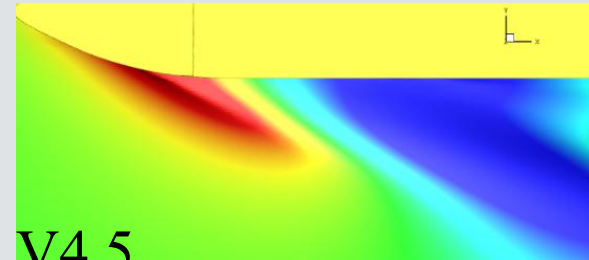
- ✓ no interface reconstruction needed, simple and easy to implement, not restricted to structured grids
- ✓ low accuracy, smeared interface, anti-diffusion or artificial steepening treatment is needed.

- *Geometric VOF method:*

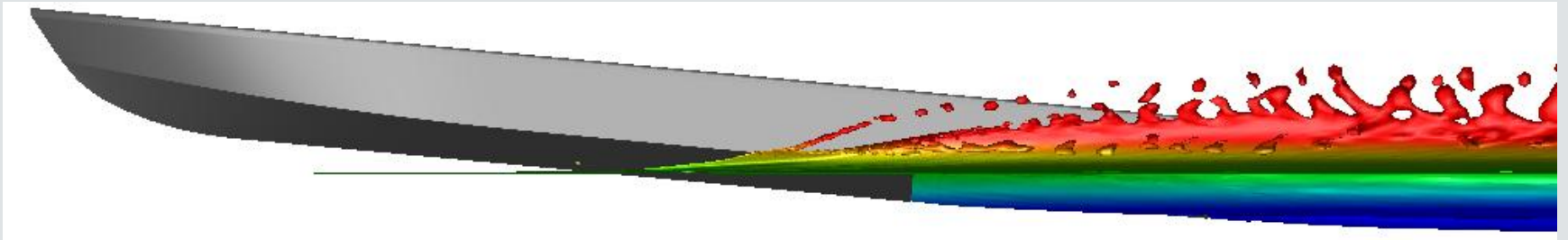
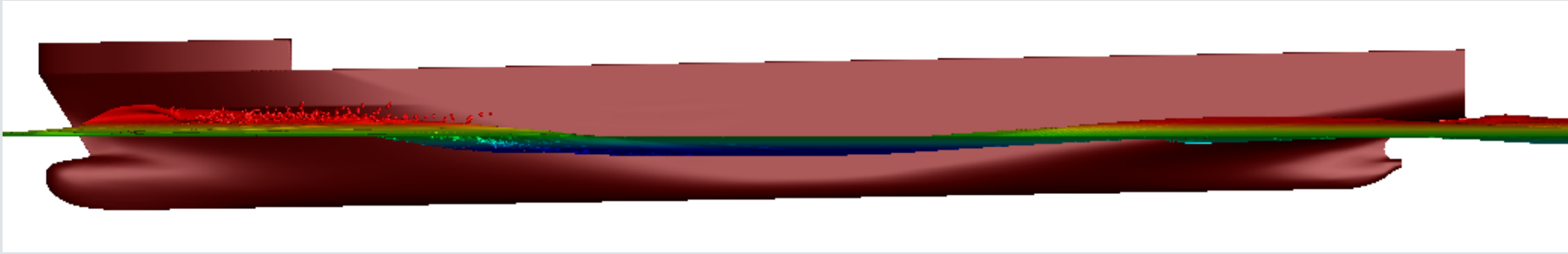
- ✓ high accuracy
- ✓ complex interface reconstruction, difficult to implement, mostly implemented on structured grids

- Most popular commercial and open-source codes use algebraic VOF method and/or with geometric method as an option: ANSYS Fluent, STAR-CCM+ , OpenFoam

- Geometric VOF method is used in FLOW-3D, Gerris (code), NFA, CFDSHIP-IOWA V6, and some in-house research codes for high-fidelity small scale low Reynolds number flows.



Interface modeling methods



Interface modeling methods

Overview of the VOF method

The interface is captured by the VOF function, F , representing the volume fraction occupied by the liquid phase in each cell:

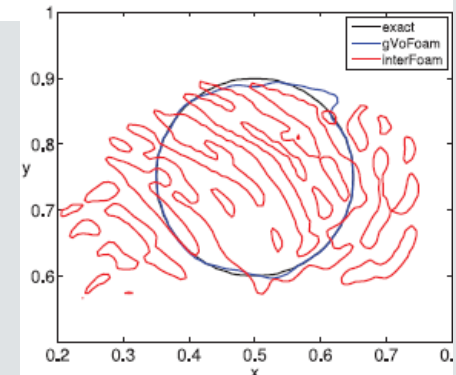
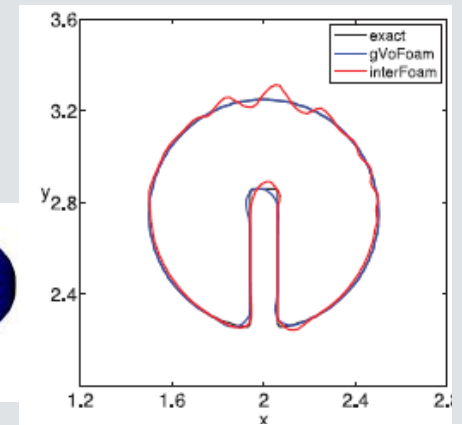
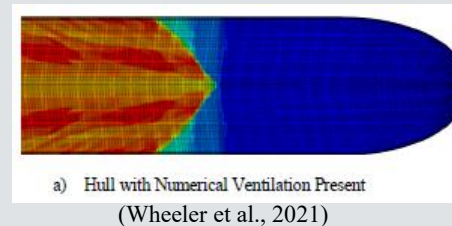
$$F(\mathbf{x}, t) = \begin{cases} 1, & \text{in the liquid,} \\ 0 < F < 1, & \text{at the interface,} \\ 0, & \text{in the gas.} \end{cases}$$

The interface position is updated by solving the VOF function advection equation,

$$\frac{\partial F}{\partial t} + \mathbf{u} \cdot \nabla F = 0.$$

Density and viscosity is treated using VOF function:

$$\rho = \rho_w F + \rho_a (1 - F), \quad \mu = \mu_w F + \mu_a (1 - F)$$



(Cifani et al., 2016, Computer & Fluids)

Implementation schemes: Geometric and Algebraic VOF methods

Geometric VOF method:

- Involving an explicit reconstruction of the interface from the VOF functions, interface advection, and flux calculation procedures

Algebraic VOF method:

- The advection equation is solved algebraically, no interface reconstruction needed,
- anti-diffusion or artificial steepening treatment is needed, such as HRIC (high resolution interface capturing) and THINC (tangent of hyperbola for interface capturing) schemes.

Interface Tracking Methods (cont'd)

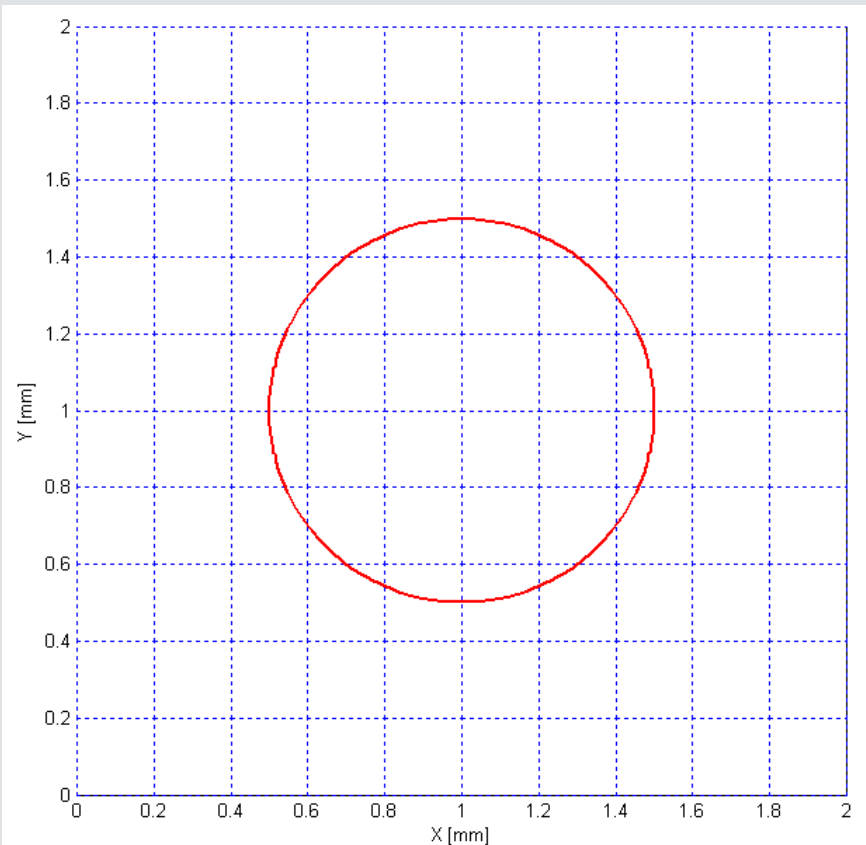
VOF method

Definition

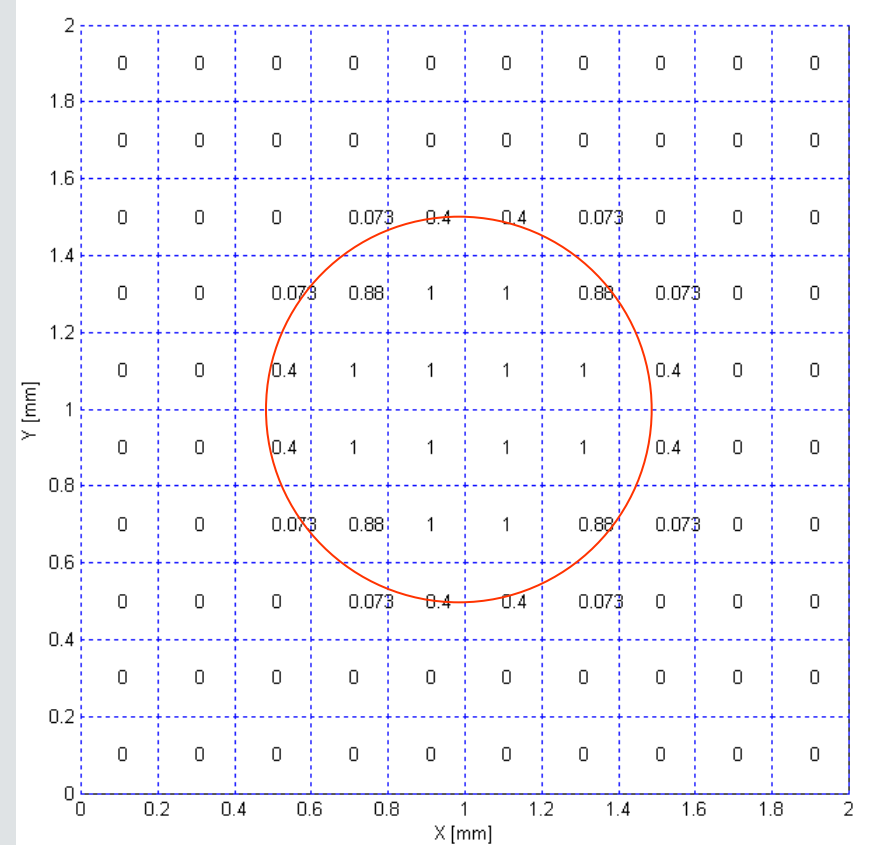
The interface is captured by the VOF function, F , representing the volume fraction occupied by the liquid phase in each cell:

$$F = \begin{cases} 1 & \text{inside the interface,} \\ 0 < F < 1 & \text{at the interface,} \\ 0 & \text{external to fluid.} \end{cases}$$

Interface Tracking Methods (cont'd)



A circle



VOF functions

Advection

The VOF function can be advected by equation,
$$\frac{DF}{Dt} = \frac{\partial F}{\partial t} + (\vec{V} \bullet \nabla)F = 0.$$

PLIC scheme is used to reconstruct the interface.

Interface Tracking Methods (cont'd)

LS method

Definition

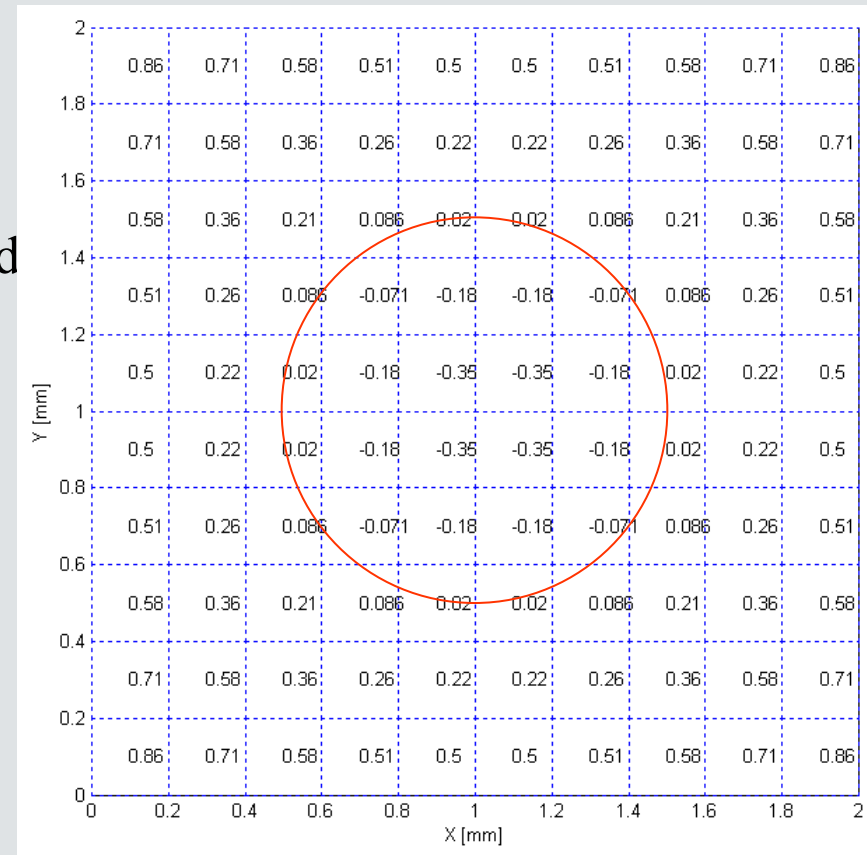
The level set function is defined as a signed distance function:

$$\begin{cases} \phi > 0 & \text{outside of the interface,} \\ \phi = 0 & \text{at the interface,} \\ \phi < 0 & \text{inside the interface.} \end{cases}$$

Advection

The level set function is evolved by

$$\frac{D\phi}{Dt} = \frac{\partial \phi}{\partial t} + (\vec{V} \bullet \nabla)\phi = 0$$



LS Functions

Interface Tracking Methods (cont'd)

To keep the LS function to be a distance function, a reinitialization process is needed :

$$\frac{\partial \phi}{\partial \tau} = \frac{\phi_0}{\sqrt{\phi_0^2 + h^2}} (1 - |\nabla \phi|)$$

ϕ_0 : levelset function at the previous step, $\phi(x,0) = \phi_0(x)$.

τ : artificial time.

CLSVOF method

Description

- Reconstruct the interface from VOF function and normal vector.
- Re-distance LS function from the reconstructed interface.
- Compute curvature and normal from LS function.

Interface Tracking Methods (cont'd)

Normal vector:

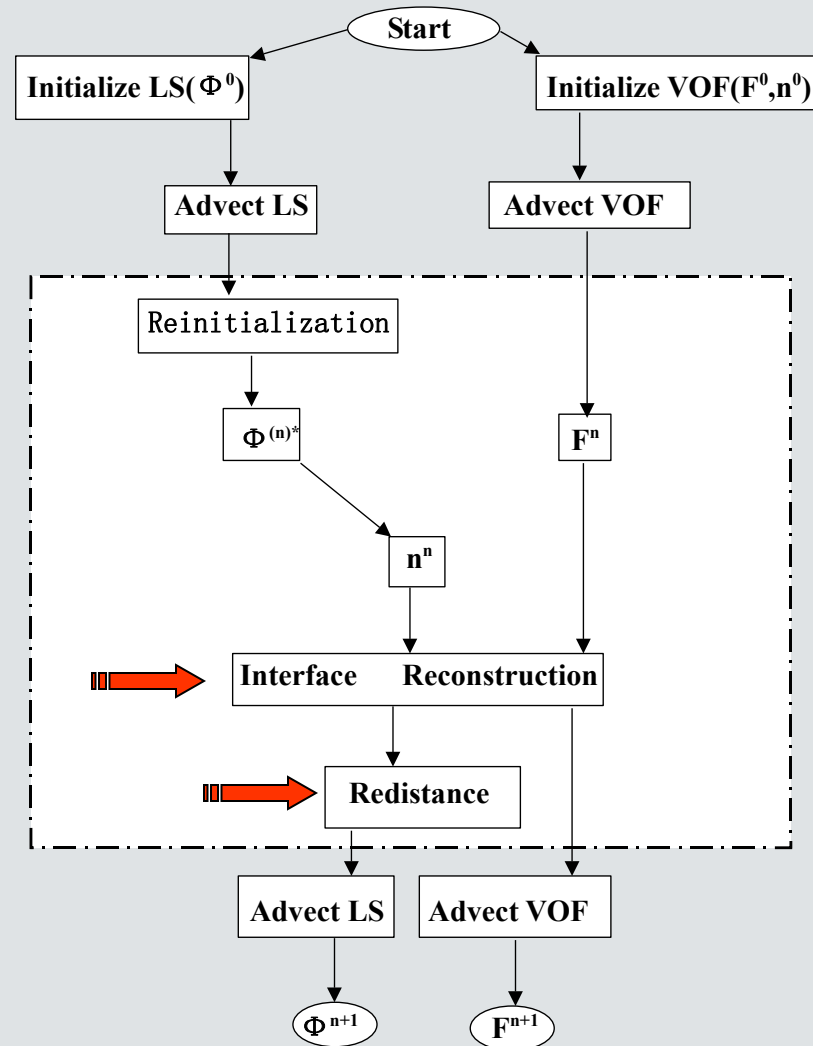
$$\vec{n} = \frac{\nabla \phi}{|\nabla \phi|} = \nabla \phi,$$

Local curvature of the interface:

$$\kappa = \nabla \bullet \nabla \phi.$$

Interface reconstruction

- Normal vector
- VOF function
- PLIC (piecewise linear interface construction) scheme:



Flow diagram for a CLSVOF method, where Φ denotes level set function, F is VOF function and n is normal. Coupled process in the dashed box.



Surface Tension Modeling

Surface tension modeling

Surface tension modeling approaches

- Continuum Surface Force (**CSF**) method

- *Calculation of surface tension force simplified (no need for interface reconstruction)*
- *No restrictions on the number, complexity, or dynamic evolution of fluid interfaces*
- *Undesirable spurious currents generated*
- *Grid convergence problems found*

A pressure boundary method (**PBM**) is developed based on the ghost fluid method (**GFM**).

Surface tension modeling (cont'd)

CSF method

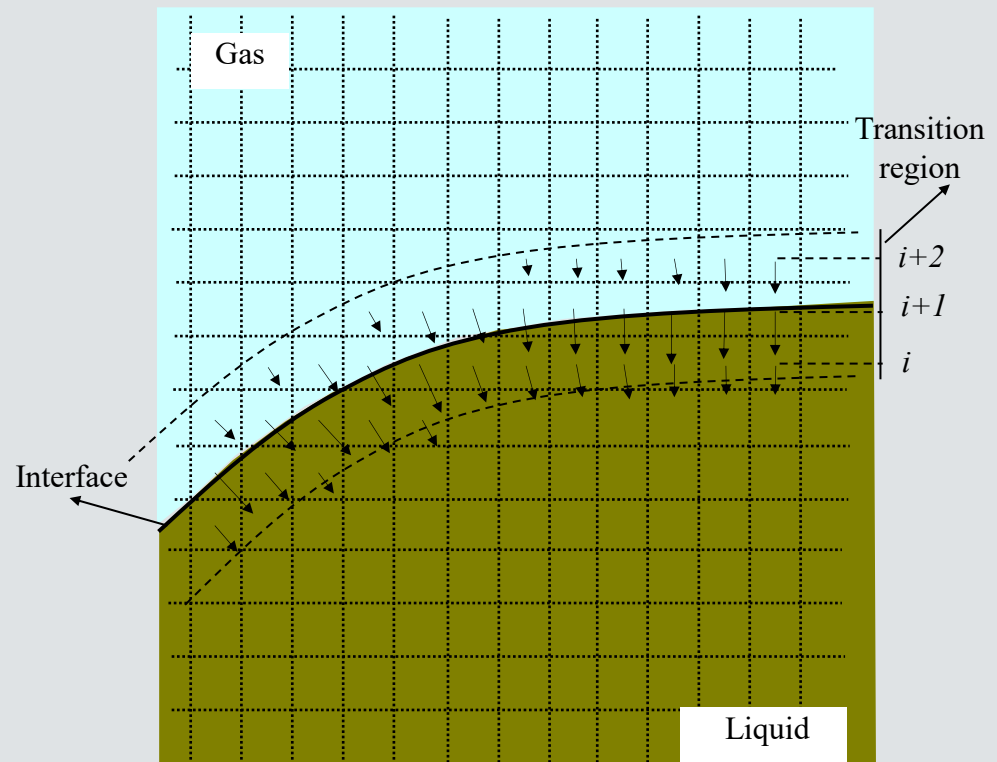
In the **CSF** model, surface tension force is treated as a body force, which acts on fluid elements lying within transition regions of finite thickness.

$$\vec{F}_b = \vec{F}_{sv} = \sigma \kappa \nabla F$$

in the VOF method,
or

$$\vec{F}_b = \vec{F}_{sv} = \sigma \kappa \vec{n} \delta(\phi)$$

in the LS method.



Surface tension forces from the CSF model

Surface tension modeling (cont'd)

To apply surface tension effect, the force $\vec{F}_b$ appears as a source term in the momentum equation,

$$\left\{ \begin{array}{l} \frac{\tilde{\vec{V}} - \vec{V}^n}{\delta t} = -\nabla \cdot (\vec{V}\vec{V})^n + \frac{1}{\rho^n} \nabla \cdot \tau^n + \vec{g}^n + \frac{1}{\rho^n} \vec{F}_b^n \\ \frac{\vec{V}^{n+1} - \tilde{\vec{V}}}{\delta t} = -\frac{1}{\rho^n} \nabla p^{n+1} \\ \nabla \cdot \vec{V}^{n+1} = 0 \end{array} \right.$$

Surface tension modeling (cont'd)

PBM method

In the **PBM** method , the surface tension effect is treated as a pressure jump condition, which is given in the Laplace's formula ,

$$p_l = \sigma \kappa + p_g$$

Then, the surface tension effect is implemented in the momentum equation via a capillary pressure gradient ,

$$\frac{\vec{V}^{n+1} - \vec{V}^n}{\delta t} = -\nabla \cdot (\vec{V}\vec{V}) - \frac{1}{\rho^n} \nabla p^{n+1} + \frac{1}{\rho^n} \nabla \cdot \tau^n + \vec{g}^n$$

where p^{n+1} is separated into two pressure terms: p_1 and p_2 , the surface tension effect is included via pressure p_1 .

Surface tension modeling (cont'd)

First, a pressure field p_1 is computed with the pressure jump conditions by the following equation,

$$\nabla \cdot \left[\frac{1}{\rho^n} \nabla p_1 \right] = 0$$

Next, the two-step projection method is applied,

$$\begin{cases} \frac{\tilde{\vec{V}} - \vec{V}^n}{\delta t} = -\nabla \cdot (\vec{V} \vec{V})^n + \frac{1}{\rho^n} \nabla \cdot \vec{\tau}^n + \vec{g}^n - \frac{1}{\rho^n} \nabla p_1 \\ \frac{\vec{V}^{n+1} - \tilde{\vec{V}}}{\delta t} = -\frac{1}{\rho^n} \nabla p_2 \end{cases}$$

$$\nabla \cdot \vec{V}^{n+1} = 0$$

where p_2 is obtained by a pressure Poisson equation,

Surface tension modeling (cont'd)

$$\nabla \bullet \left[\frac{1}{\rho^n} \nabla p_2 \right] = \frac{\nabla \bullet \tilde{\vec{V}}}{\delta t}$$

Finally, pressure p^{n+1} is computed using equation

$$p^{n+1} = p_1 + p_2$$

Surface tension modeling (cont'd)

Discretization of the pressure equation

$$\nabla \cdot \left[\frac{1}{\rho} \nabla p_1 \right]_{i,j} = 0$$

$$\frac{(\nabla p_1 / \rho)_{i+1/2,j} - (\nabla p_1 / \rho)_{i-1/2,j}}{\Delta x} + \frac{(\nabla p_1 / \rho)_{i,j+1/2} - (\nabla p_1 / \rho)_{i,j-1/2}}{\Delta y} = 0$$

Free surface flows

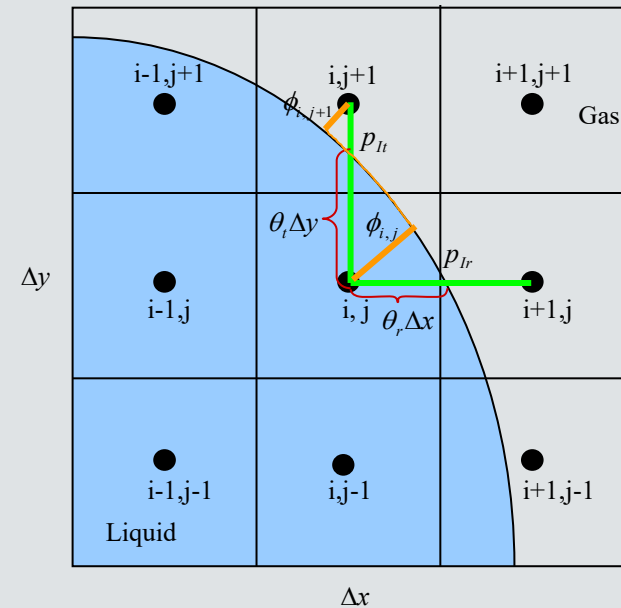
The pressure gradients at faces are discretized as

$$(\nabla p)_{i-1/2,j} = \frac{p_{i,j} - p_{i-1,j}}{\Delta x}$$

$$(\nabla p)_{i+1/2,j} = \frac{p_{Ir} - p_{i,j}}{\theta_r \Delta x}$$

$$(\nabla p)_{i,j-1/2} = \frac{p_{i,j} - p_{i,j-1}}{\Delta y}$$

$$(\nabla p)_{i,j+1/2} = \frac{p_{It} - p_{i,j}}{\theta_t \Delta y}$$



Determination of the distance from cell center to the interface in x/y direction and pressure discretization at the interface

Surface tension modeling (cont'd)

The pressure at the interface is calculated by

$$p_{Ir} = p_g + \sigma \kappa_{Ir}$$

$$p_{It} = p_g + \sigma \kappa_{It}$$

The curvature at the interface is calculated by

$$\kappa_{Ir} = \theta_r \kappa_{i+1,j} + (1 - \theta_r) \kappa_{i,j}$$

$$\kappa_{It} = \theta_t \kappa_{i,j+1} + (1 - \theta_t) \kappa_{i,j}$$

Define θ to be the fraction of the distance from the cell (i,j) center to the interface:

$$\theta_r = \frac{|\phi_{i,j}|}{|\phi_{i,j}| + |\phi_{i+1,j}|}$$

$$\theta_t = \frac{|\phi_{i,j}|}{|\phi_{i,j}| + |\phi_{i,j+1}|}$$

Surface tension modeling (cont'd)

Two-phase flows

At the single phase, pressure gradient is discretized using the standard method,

$$(\nabla p)_{i-1/2,j} = \frac{p_{i,j} - p_{i-1,j}}{\Delta x}$$

Pressure jump condition at the interface,

$$p_{ll} - p_{lg} = \sigma \kappa$$

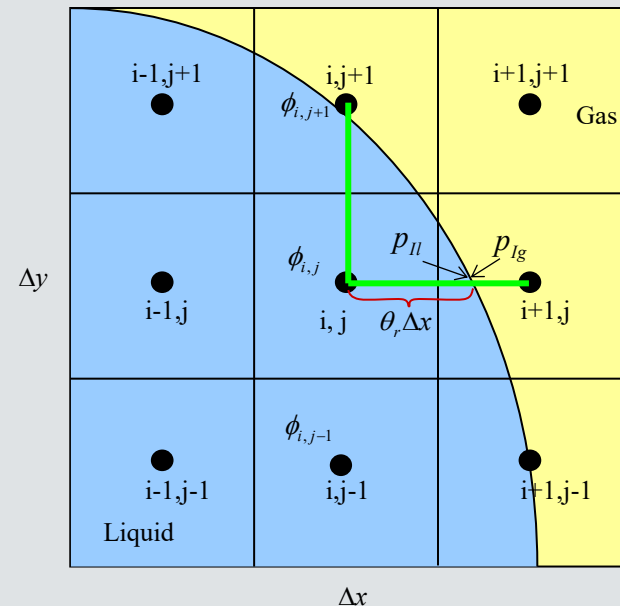
At the right face of cell (i, j),

$$(\nabla p)_{i+1/2,j} = \frac{p_{ll} - p_{i,j}}{\theta_r \Delta x}$$

$$(\nabla p)_{i+1/2,j} = \frac{(p_{lg} + \sigma \kappa) - p_{i,j}}{\theta_r \Delta x}$$

p_{lg} can be extrapolated, here it is simply approximated by,

$$p_{lg} \approx p_{i+1,j}$$



(a)

(a) Right face in the liquid phase

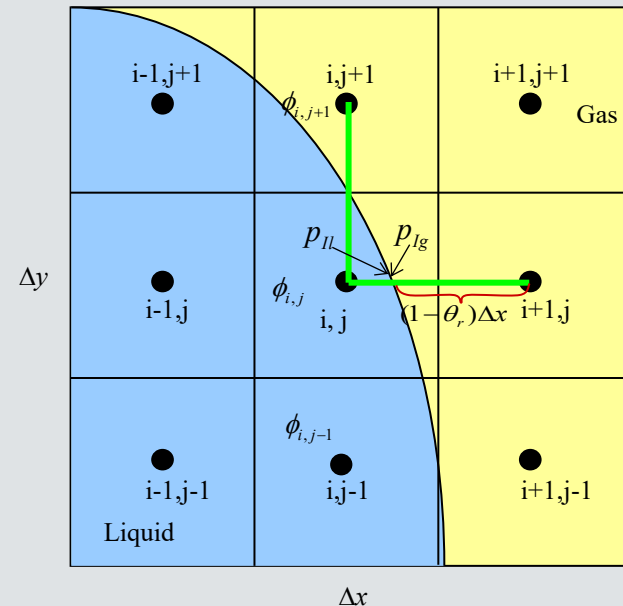
Surface tension modeling (cont'd)

$$(\nabla p)_{i+1/2,j} = \frac{(p_{i+1,j} + \sigma\kappa) - p_{i,j}}{\theta_r \Delta x}$$

For case b, pressure gradient will be discretized as,

$$(\nabla p)_{i+1/2,j} = \frac{p_{i+1,j} - p_{Ig}}{(1 - \theta_r) \Delta x}$$

$$(\nabla p)_{i+1/2,j} = \frac{p_{i+1,j} - (p_{ll} - \sigma\kappa)}{(1 - \theta_r) \Delta x}$$



(b)

(b) Right face in the gas phase

$$(\nabla p)_{i+1/2,j} = \frac{p_{i+1,j} - (p_{i,j} - \sigma\kappa)}{(1 - \theta_r) \Delta x}$$

Surface tension modeling (cont'd)

Substitute pressure gradients into the following equation,

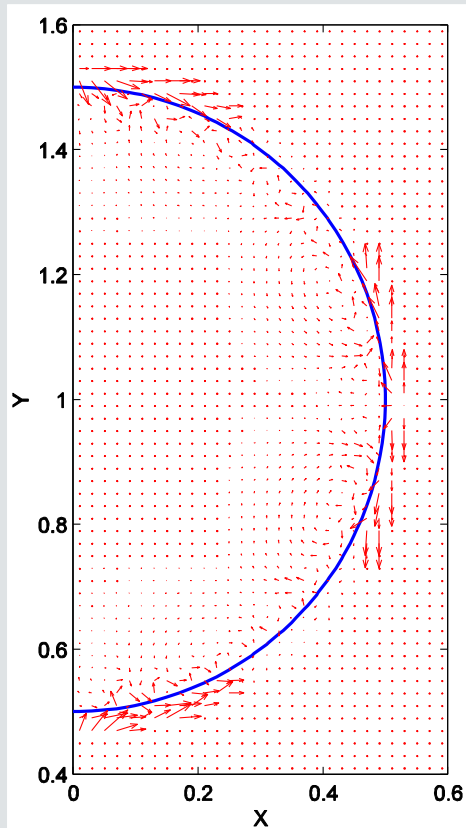
$$\nabla \cdot \left[\frac{1}{\rho} \nabla p_1 \right]_{i,j} = \frac{(\nabla p_1 / \rho)_{i+1/2,j} - (\nabla p_1 / \rho)_{i-1/2,j}}{\Delta x} + \frac{(\nabla p_1 / \rho)_{i,j+1/2} - (\nabla p_1 / \rho)_{i,j-1/2}}{\Delta y} = 0$$

a **symmetric discretization** of the pressure equation is obtained, the resulting symmetric linear system can be readily solved using an incomplete Cholesky conjugate gradient (ICCG) method.

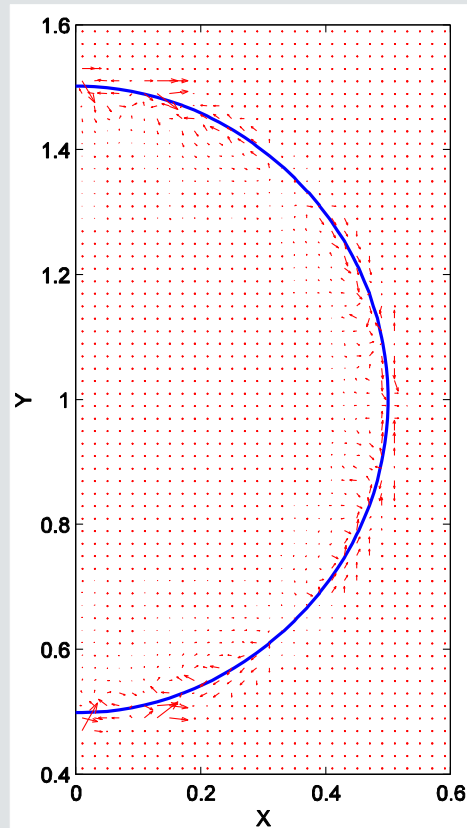
Surface tension modeling (cont'd)

Numerical tests

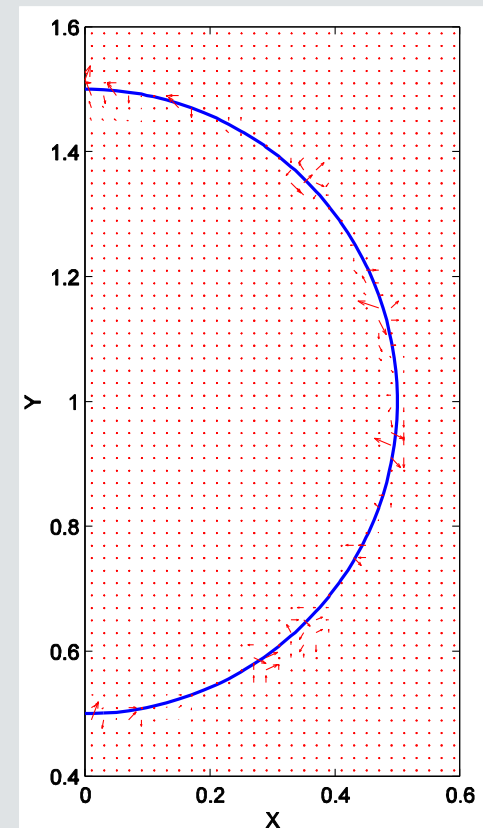
Free surface flows: Static test



(a) *VOF - CSF*



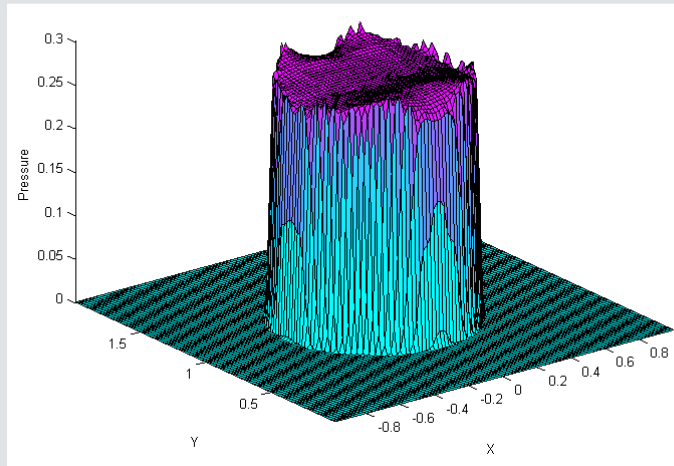
(b) *CLSVOF - CSF*



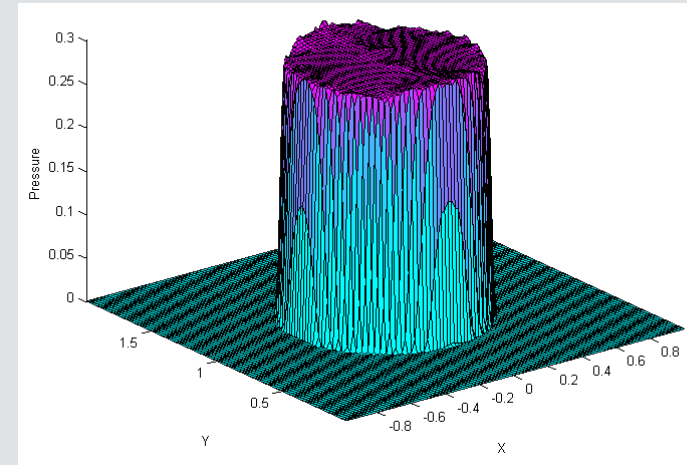
(c) *CLSVOF - PBM*

Velocity profiles for a stationary droplet in air, after 0.5ms, grid:0.02X0.02mm.

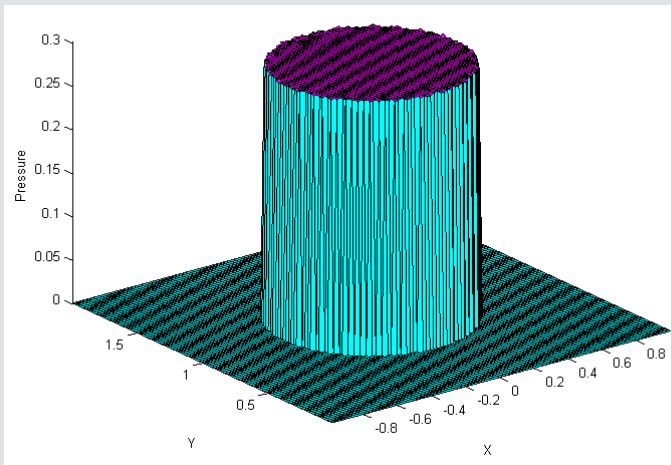
Surface tension modeling (cont'd)



(a) *VOF – CSF*



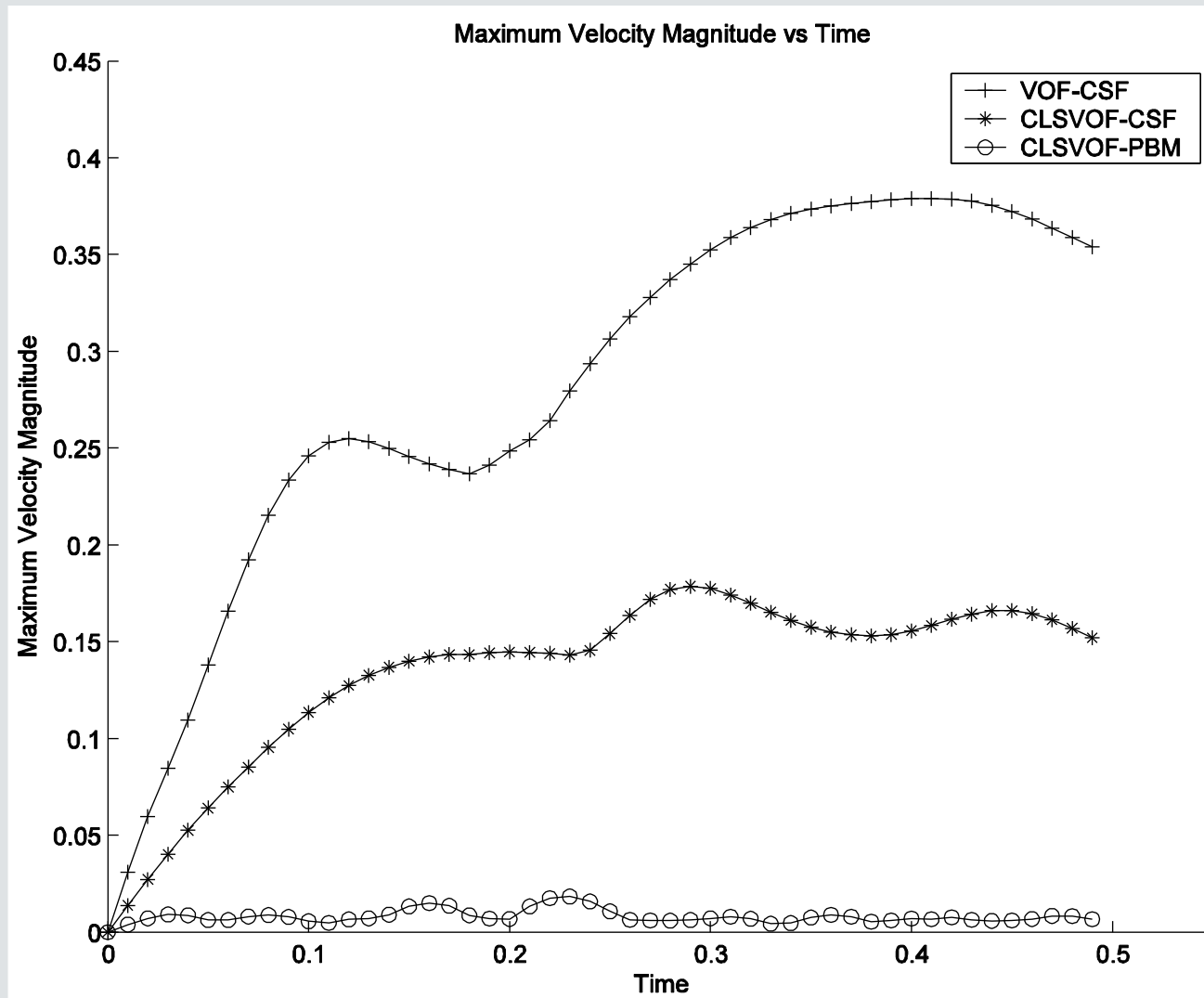
(b) *CLSVOF – CSF*



(c) *CLSVOF – PBM*

Pressure distribution for a stationary droplet in air, at time 0.5ms, grid: 0.02mmX0.02mm

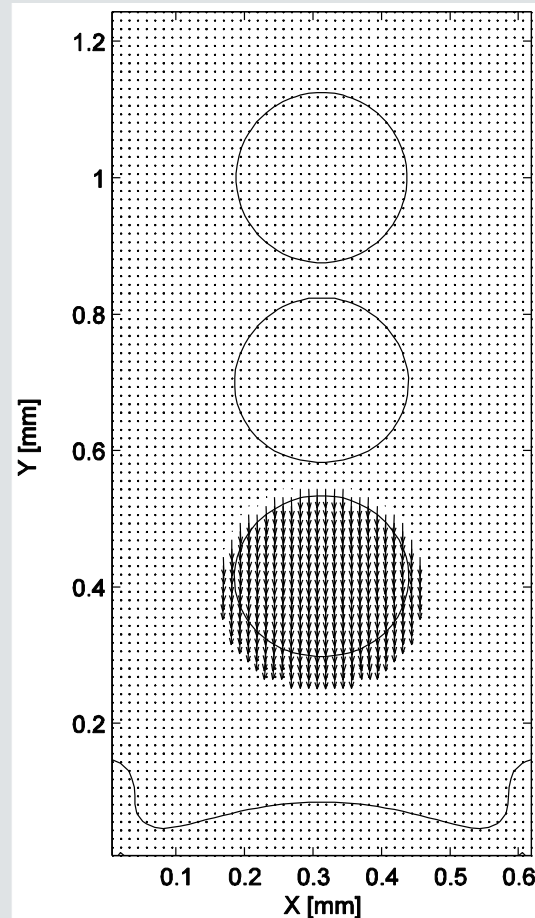
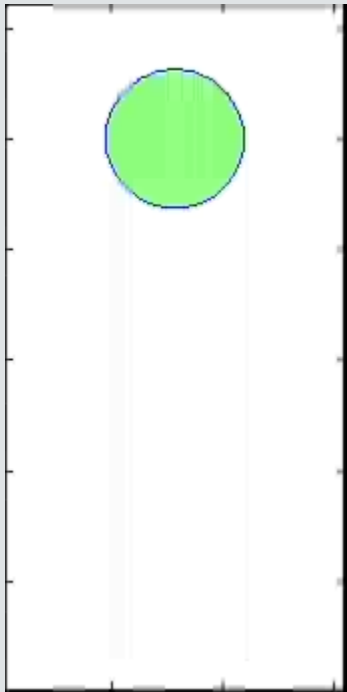
Surface tension modeling (cont'd)



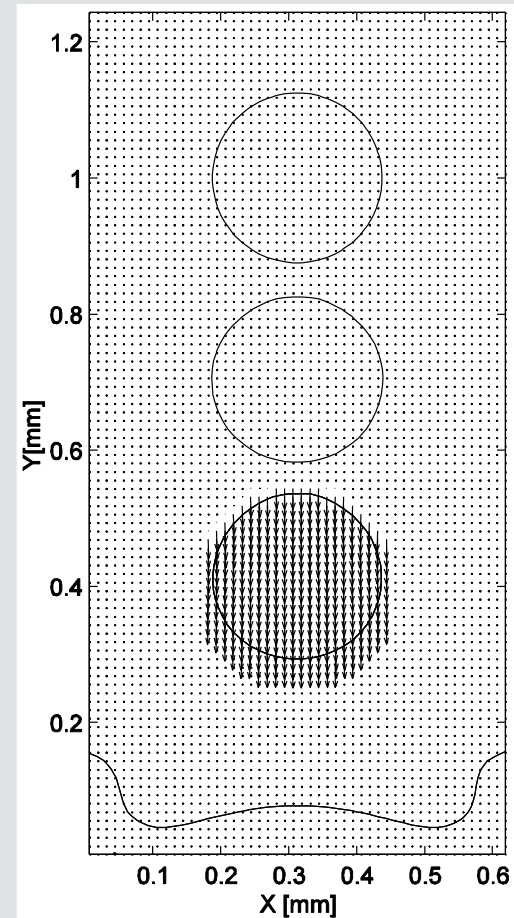
Maximum velocity magnitude variation with time. Grid:0.04mmX0.04mm.

Surface tension modeling (cont'd)

Free surface flows: A falling droplet in air



(a) CLSVOF – CSF

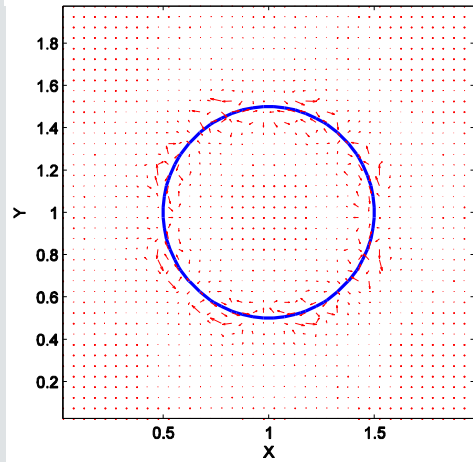
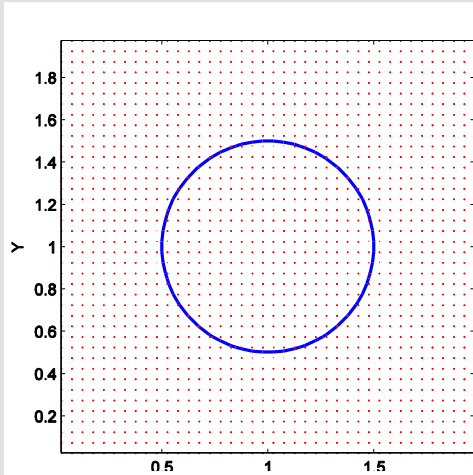


(b) CLSVOF – PBM

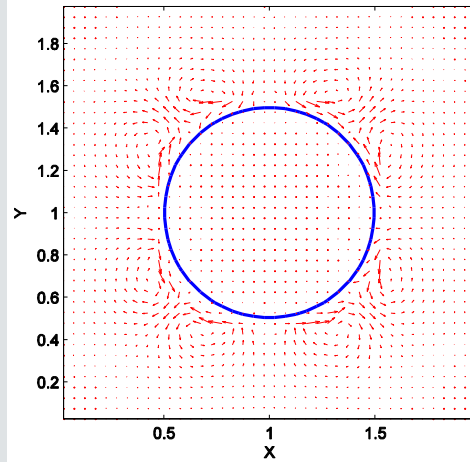
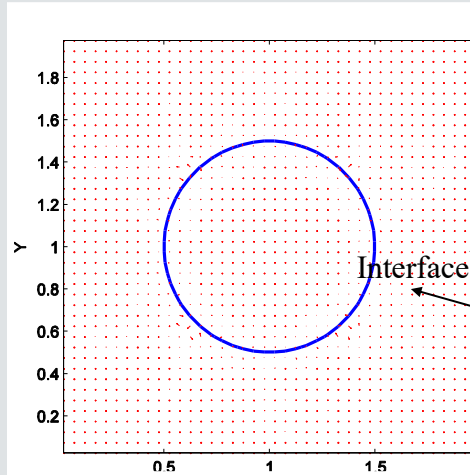
Time sequence and velocity profiles for a falling droplet in air

Surface tension modeling

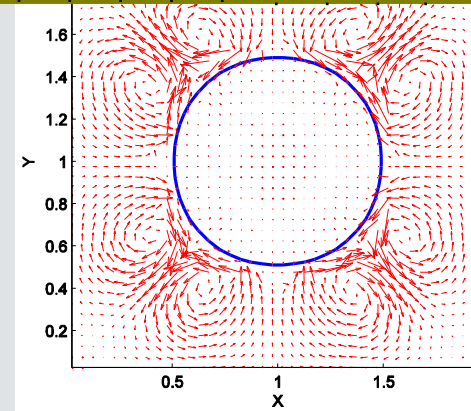
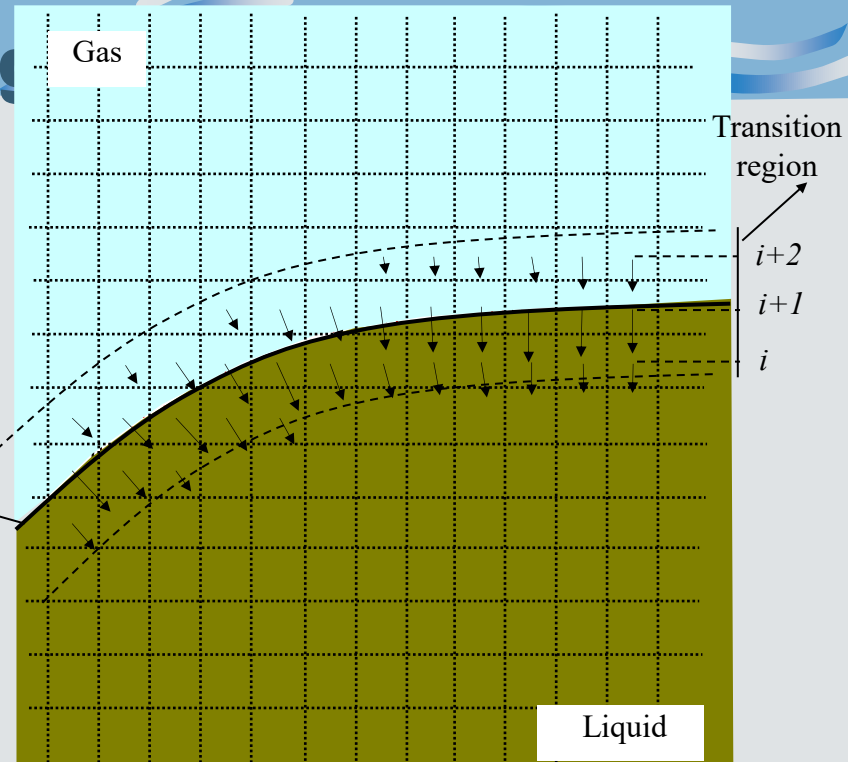
Two phase flows: Spurious currents test



Density ratio: 1



100

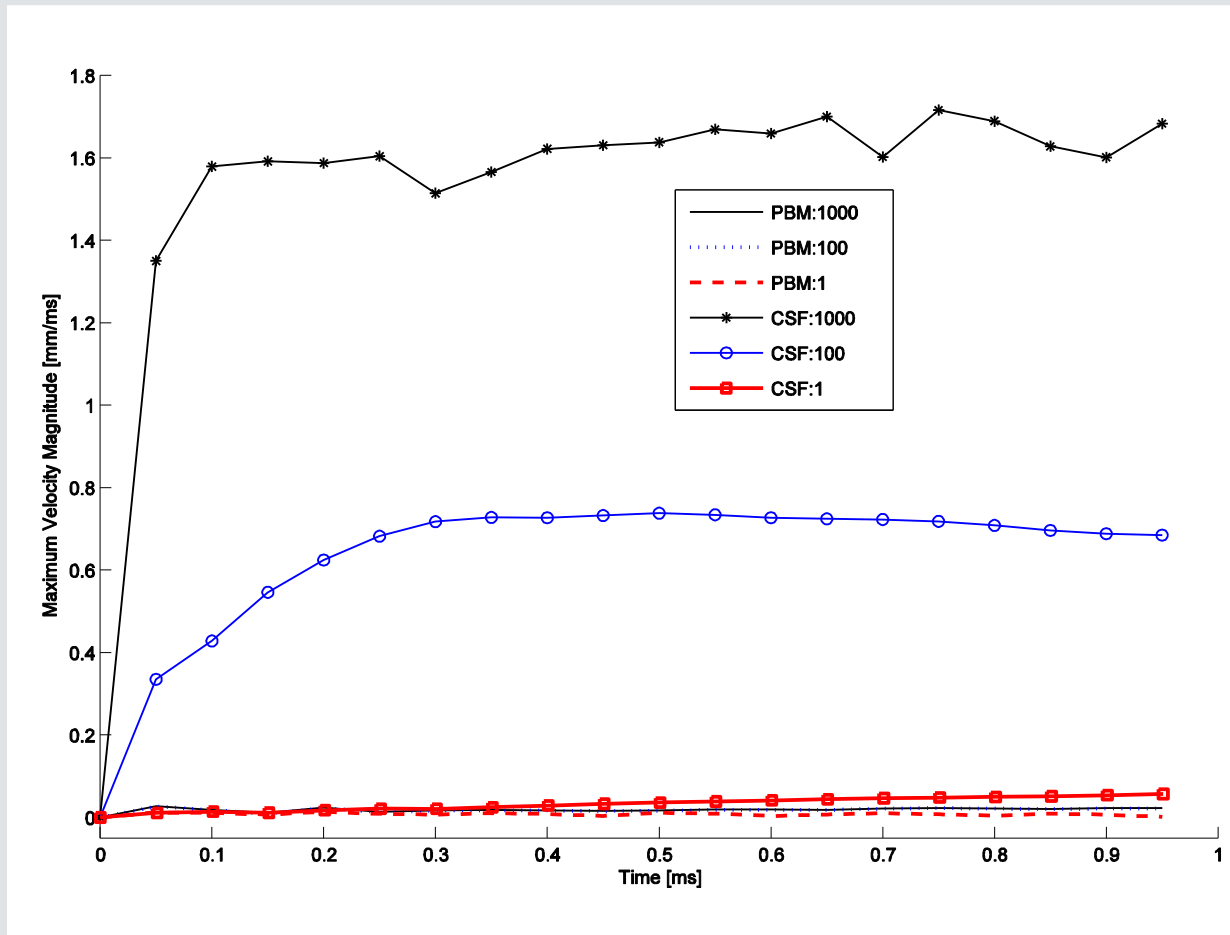


1000

CSF

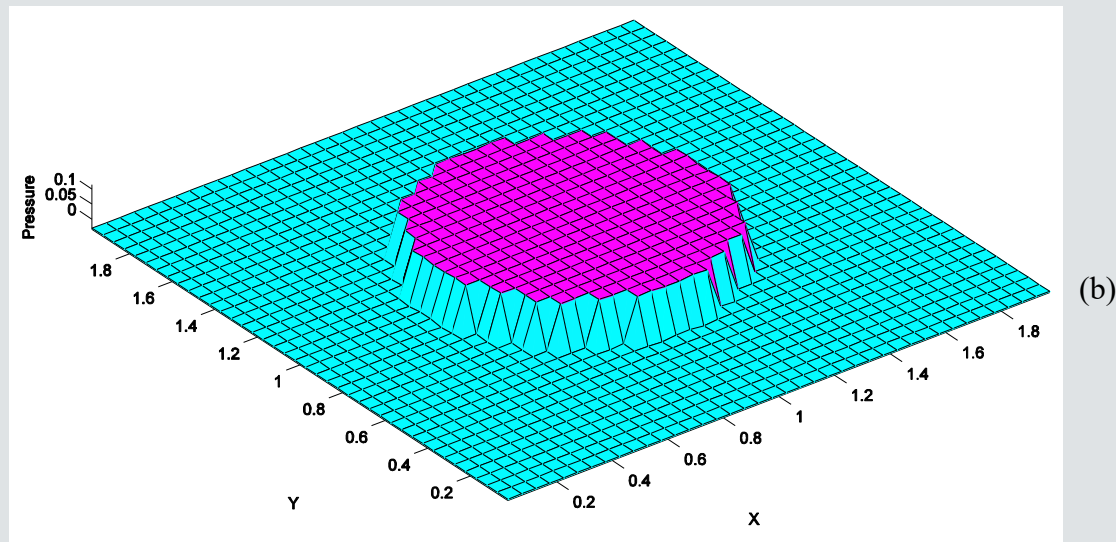
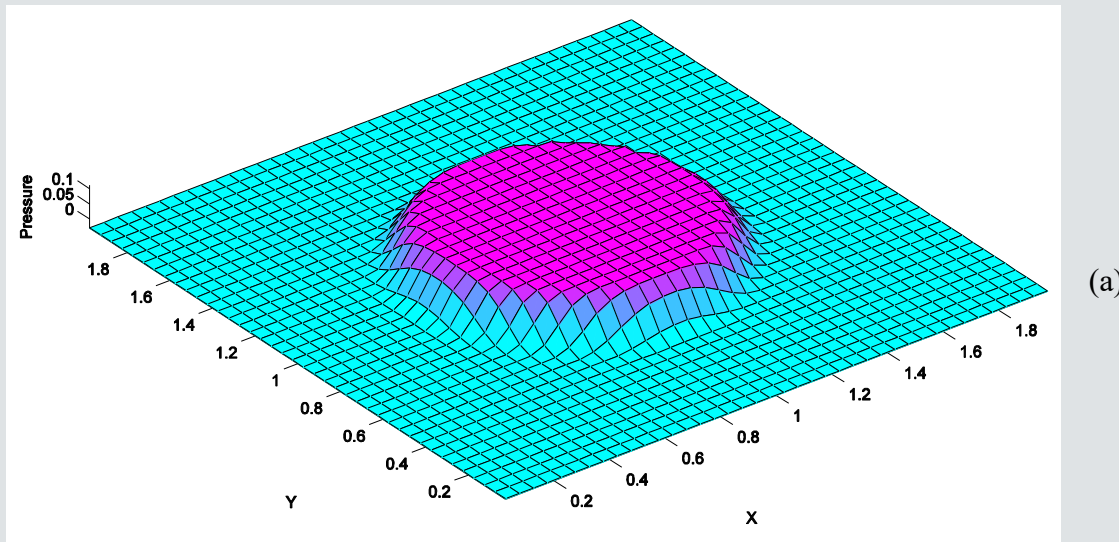
Velocity profiles for a stationary droplet for cases with different density ratios.

Surface tension modeling (cont'd)



Maximum velocity magnitude versus time for different density ratios.

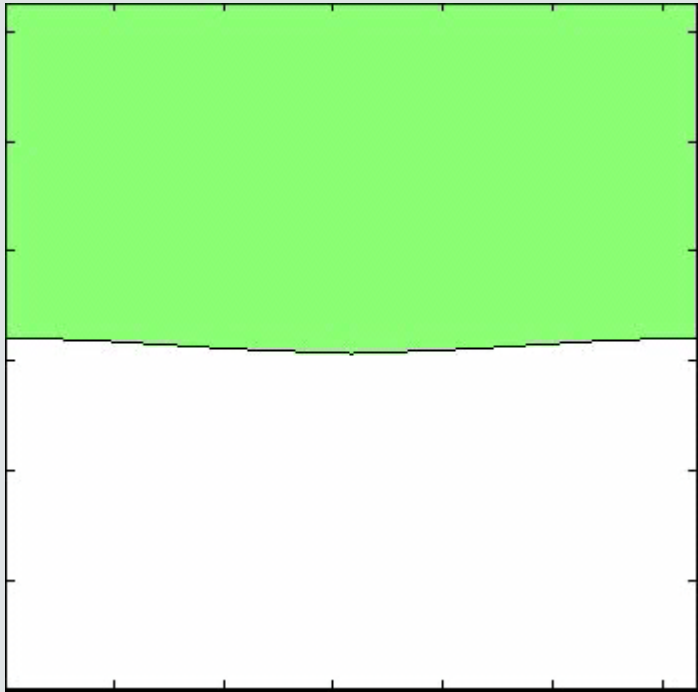
Surface tension modeling (cont'd)



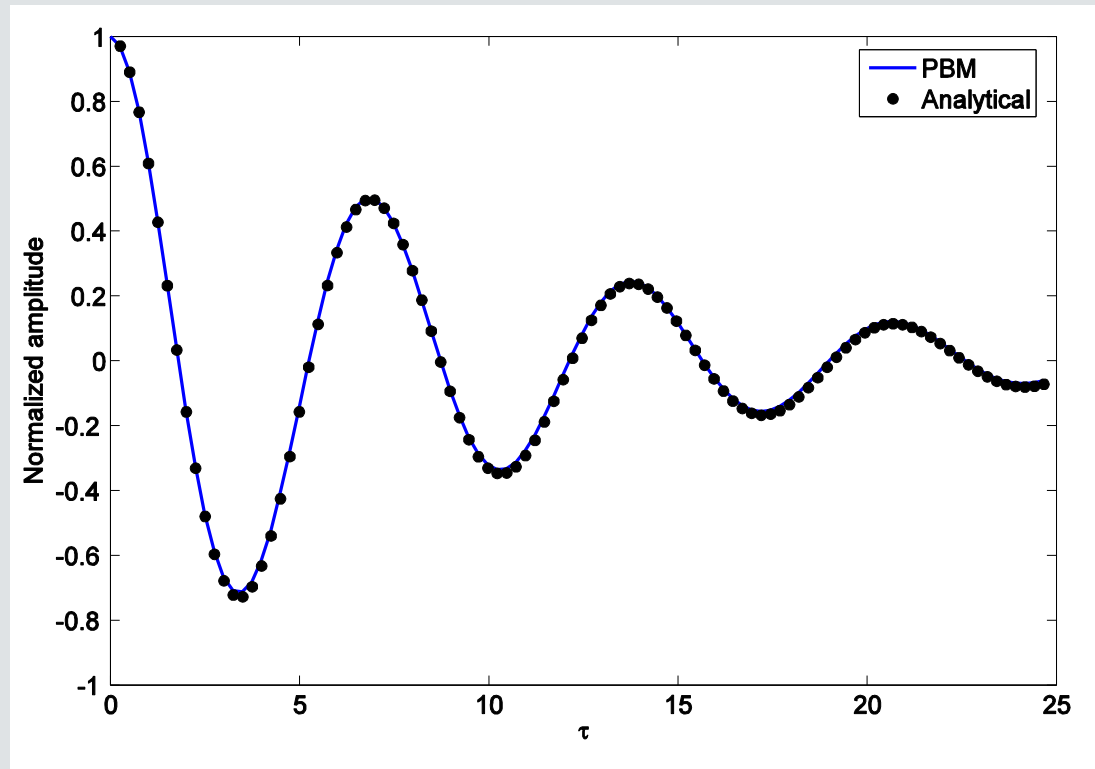
Pressure distribution: (a) CSF method; (b) PBM method.

Surface tension modeling (cont'd)

Two phase flows: Capillary wave



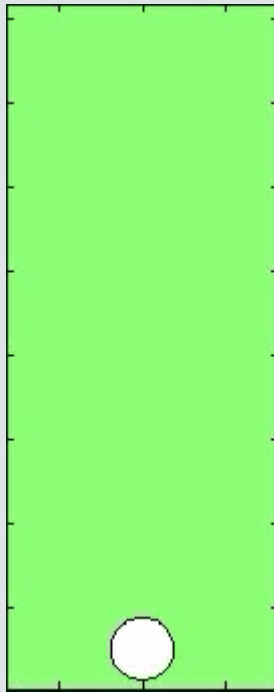
Oscillation of an interface between two viscous fluids



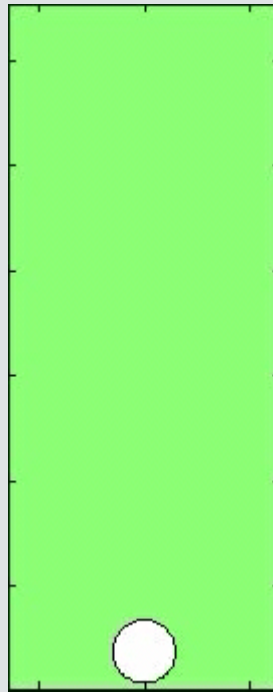
Time evolution of the amplitude of the capillary wave.
Analytical solution, [Prosperetti, 1981]

Surface tension modeling (cont'd)

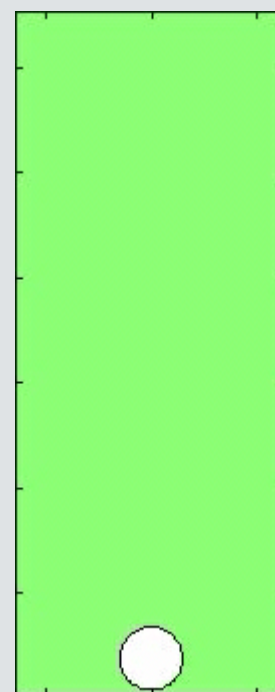
Two phase flows: air bubbles in water



(a) $d=1.5\text{mm}$



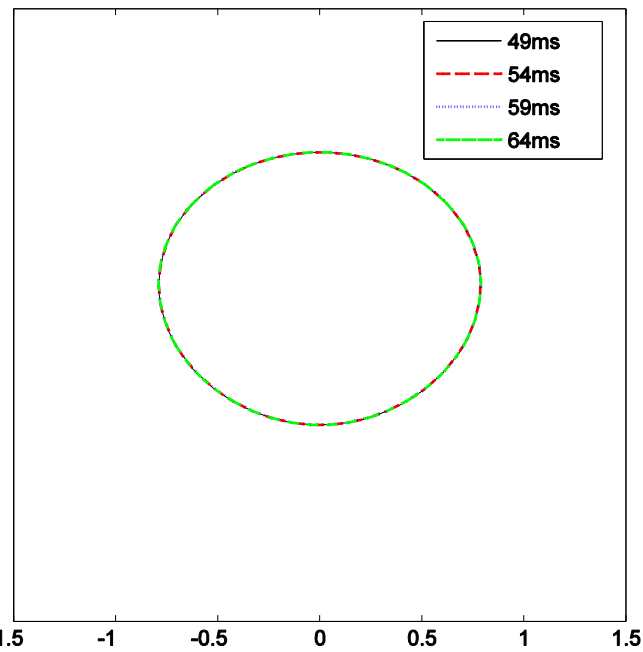
(b) $d=3.0\text{mm}$



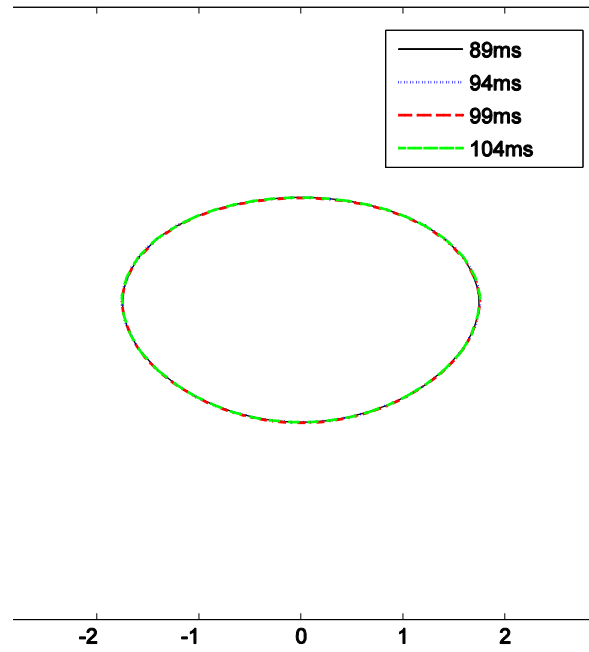
(c) $d=6.0\text{mm}$

Surface tension modeling (cont'd)

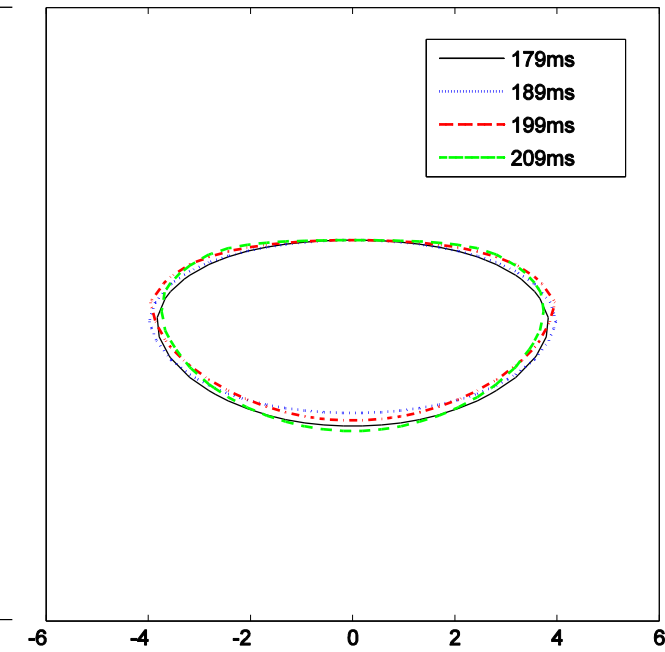
Two phase flows: air bubbles in water



(a) $d=1.5\text{mm}$



(b) $d=3.0\text{mm}$



(c) $d=6.0\text{mm}$

Interface modeling methods in CFDShip-Iowa V5.5

Wang, Z, Stern, F. Volume-of-fluid based two-phase flow methods on structured multiblock and overset grids. Int J Numer Meth Fluids. 2022; 94(6): 557– 582. doi:10.1002/fld.5066.

- **A geometric VOF method is implemented on the static/dynamic overset grids with a new VOF interpolation algorithm developed.**

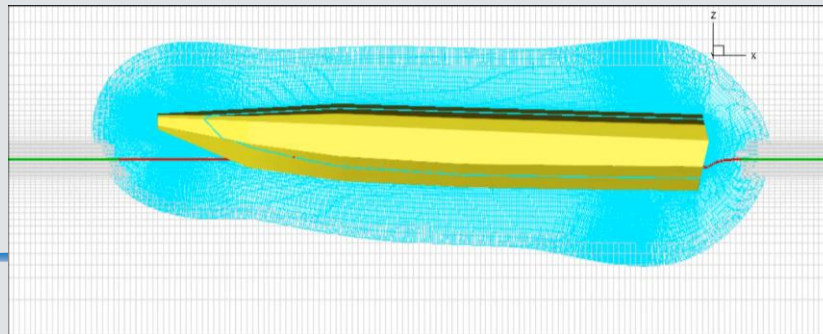
The VOF value cannot be directly interpolated through the overset grid boundaries since the VOF function is a discontinuous step function.

The VOF value in the fringe cells is calculated geometrically based on the reconstructed interface.

- **The gravity and surface tension forces are implemented using a VOF based ghost fluid method,**
A ghost pressure interpolation scheme is developed to handle the pressure jump across the interface due to the density changes and surface tension effect.

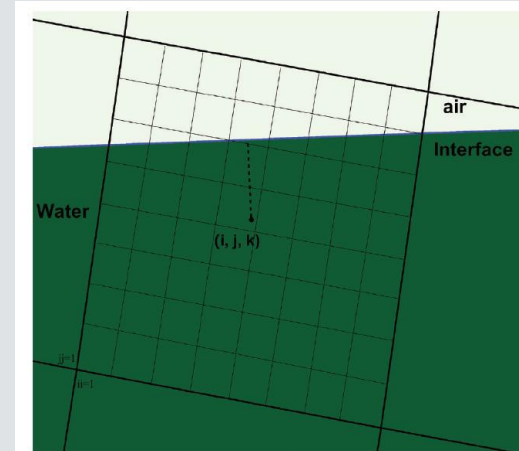
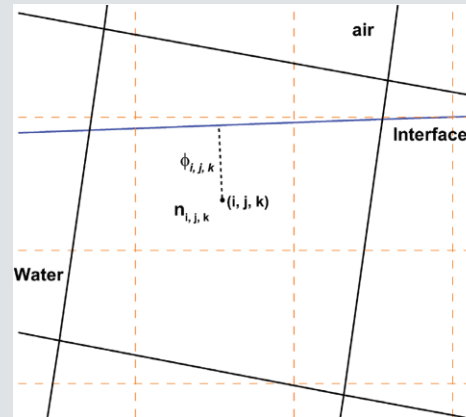
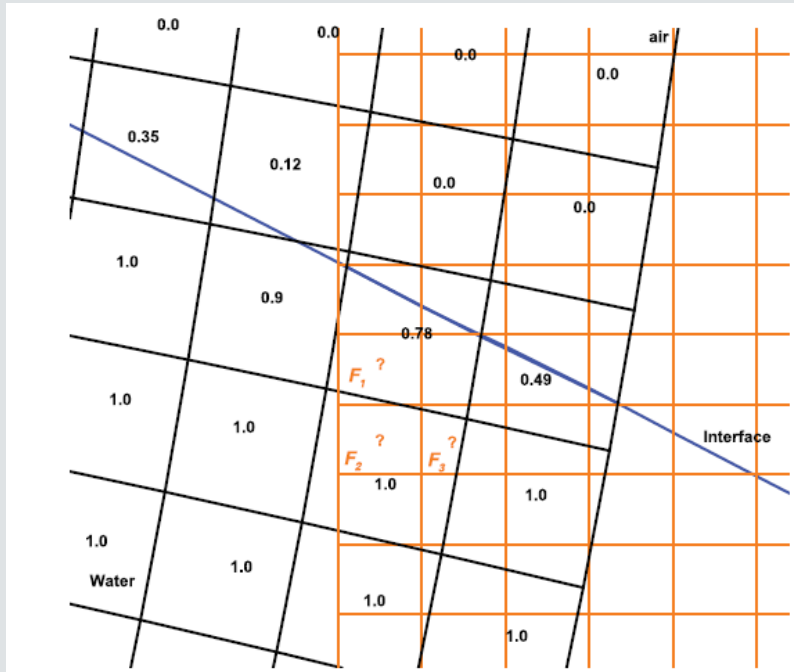
- **No-slip boundary condition and moving contact line problem.**

A blanking distance (d_{blank}) from the wall is used for the interface functions chosen based on the y^+ values. The numerical experiments show that the blank distance needs to satisfy $30 < y^+ < 200$ in consideration of accuracy and stability, and a value of $y^+ \sim 100$ is usually used in practice



Interface modeling methods

VOF interpolation on overset grid



$$ax + by + cz = d$$

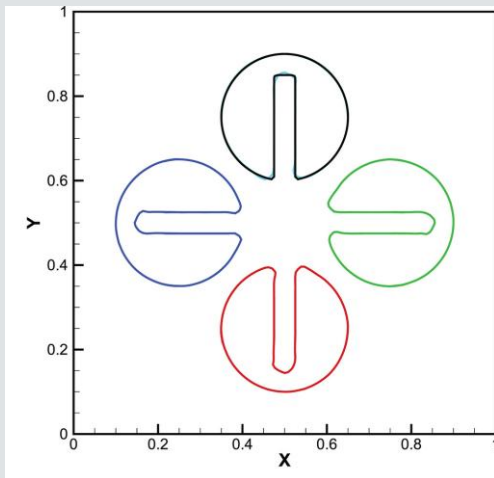
$$d = \phi_{i,j,k} + ax_{i,j,k} + by_{i,j,k} + cz_{i,j,k}$$

$$F = Vol_{water} / Vol_{total}$$

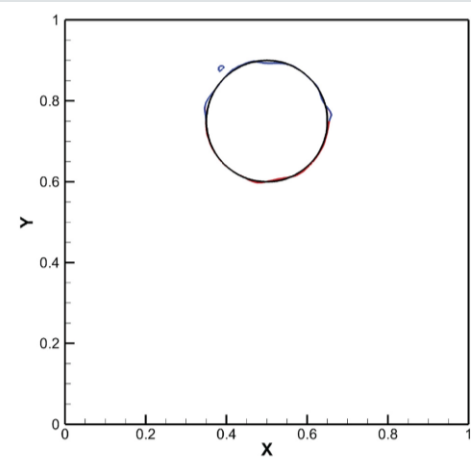
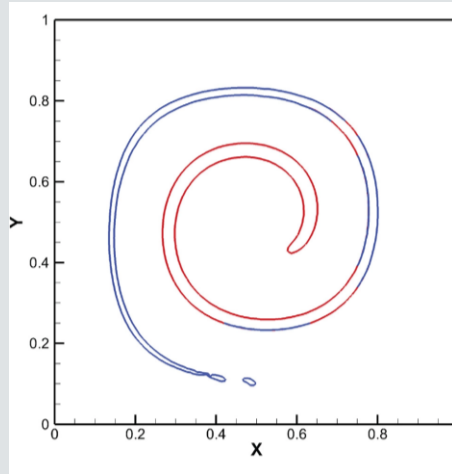
Interface modeling methods

VOF on overset grid tests

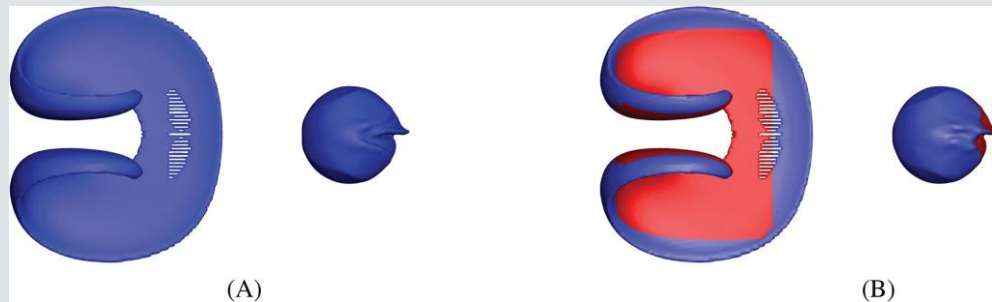
Slotted disk



Single vertex flow



3D deformation test



Interface modeling methods

Gravity and surface tension forces implementation

Body forces including the **interfacial force** (such as the surface tension force) and the **volumetric forces** (such as the gravity and electromagnetic forces), greatly depends on the interface tracking method, variable arrangement on the grid, and pressure velocity coupling.

$$\rho \left(\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} \right) = - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \rho g_i + F_{st}$$

Jump condition for piezometric pressure

$$\hat{p}_{water} - \hat{p}_{air} = (\rho_{water} - \rho_{air})gz_I.$$

$$\hat{p}_B^g = a\hat{p}_B + (1-a)\hat{p}_P - ad.$$

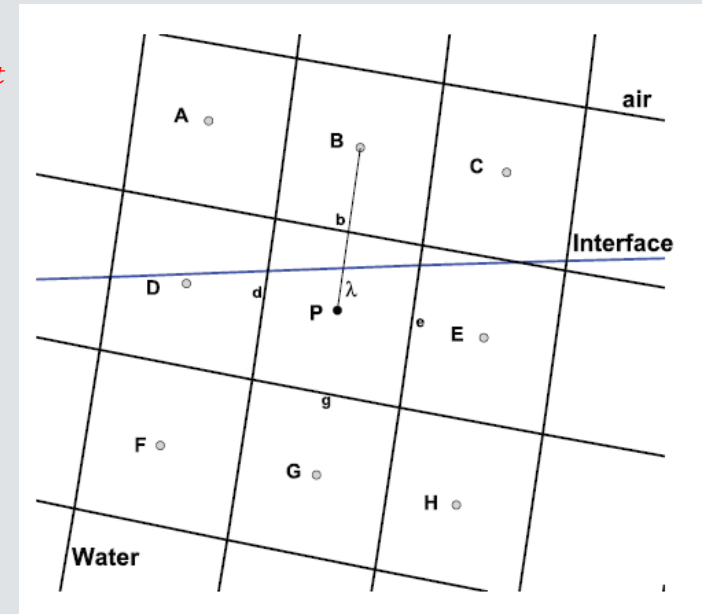
Jump condition due to surface tension

$$p_{water} - p_{air} = \sigma \kappa_I,$$

$$p_B^g = ap_B + (1-a)p_P + \rho_P \Delta z g + a\sigma \kappa_I.$$

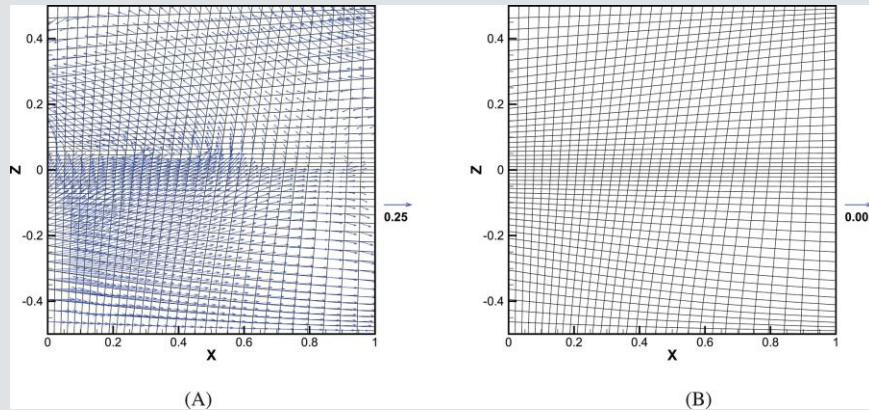
$$a = \frac{\frac{1}{\rho_B}}{0.5\left(\frac{1}{\rho_P} + \frac{1}{\rho_B}\right)},$$

$$d = 0.5(z_P + z_B)(\rho_B - \rho_P)g.$$

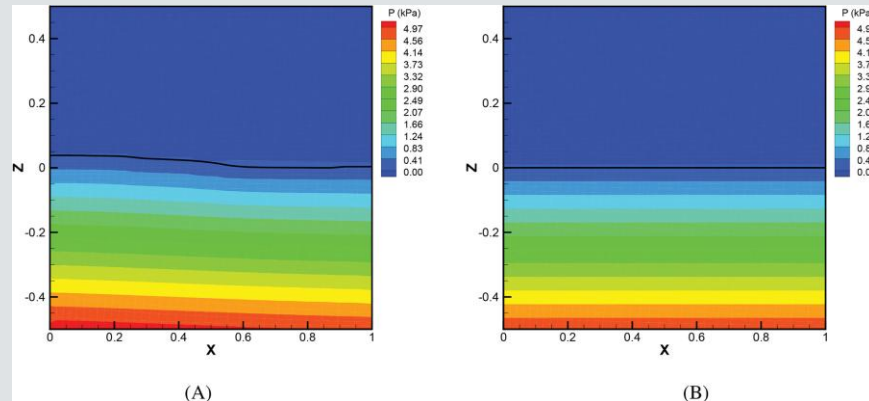


Interface modeling methods

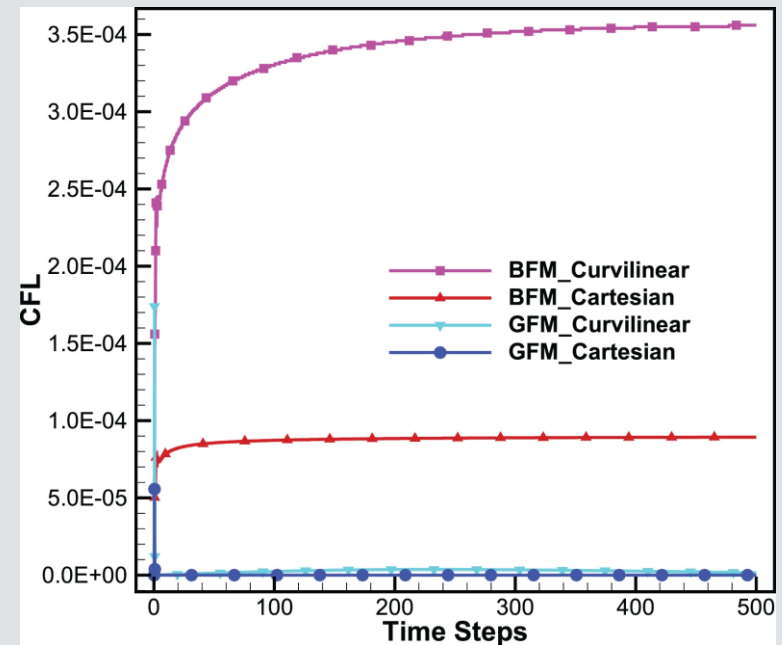
Hydrostatic pressure test for gravity



Velocity profiles for the hydro-static test on general structured grid: (A) body force method; (B) VOF ghost pressure method



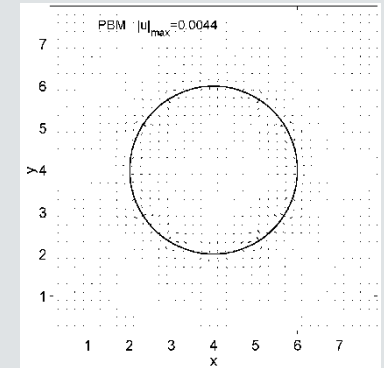
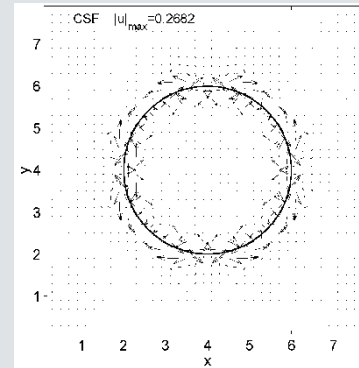
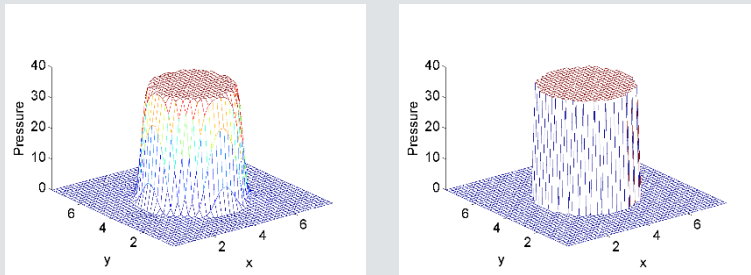
Pressure distribution and interface profiles for the hydro-static test on general structured grid: (A) body force method; (B) VOF ghost pressure method



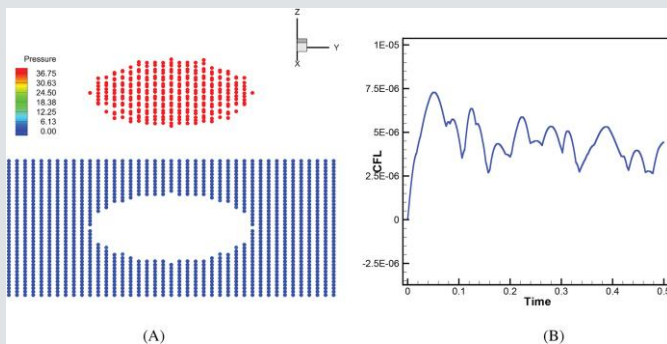
Comparison of the CFL numbers for BFM and GFM methods

Interface modeling methods

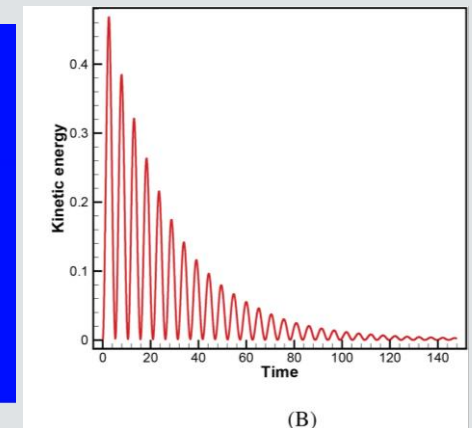
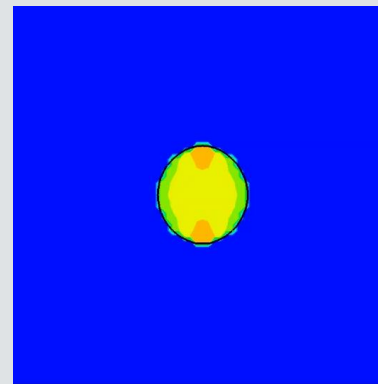
Static circular and oscillating elliptical drops for surface tension



Pressure distribution and velocity fields of a circular droplet (Wang and Tong, 2009)



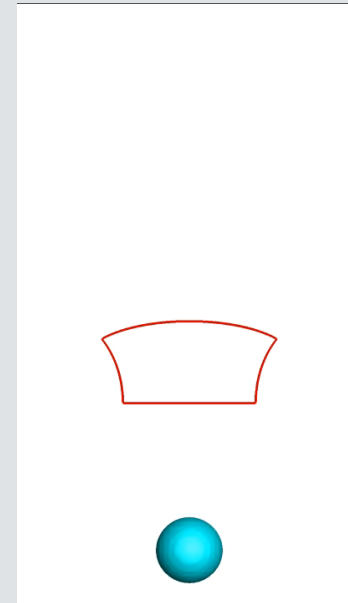
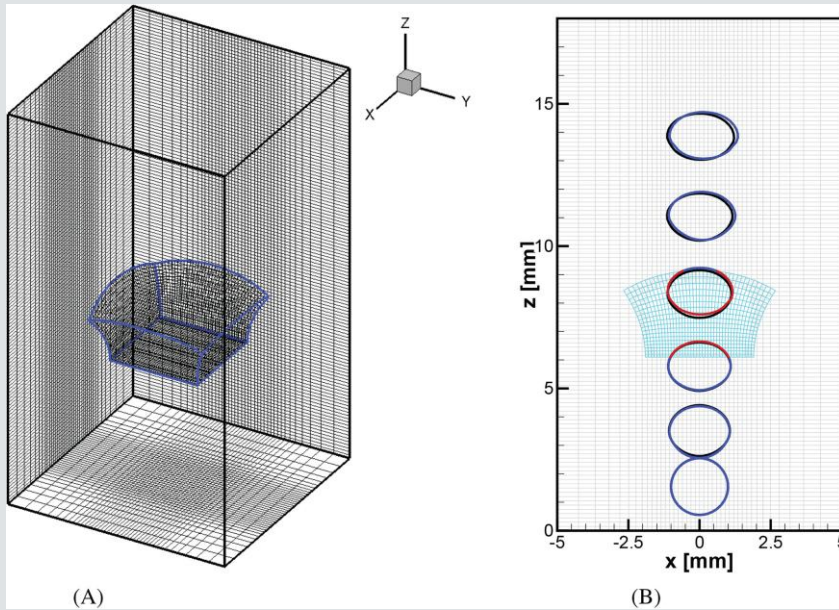
(A) Pressure distribution for a circular static drop. (B) Time history of the CFL number for a circular static drop



(A) Pressure distribution for an oscillating drop. (B) Time history of kinetic energy for an oscillating drop

Interface modeling methods

Air bubble rising in quiescent water



(A) Overset grid layout; and (B) time sequences for an air bubble rising in quiescent water

The computed terminal velocity is 0.291 mm/ms which is in excellent agreement with the experimental value of 0.287 mm/ms calculated using the experimental correlation.

$$u_T = \sqrt{2\sigma / \rho_l d_e + \Delta\rho g d_e / 2}$$