Tackling Turbulence using Large Eddy Simulation

July 2003

by
Tony Saad

Advised by
Dr. Michel El Hayek

http://tsaad.utsi.edu - tsaad@utsi.edu

Contents

- The Problem
- Equations of Fluid Motion
- Turbulence Models
- Large Eddy Simulation
- Numerical Methods
- Boundary Conditions
- Code Validation

The Problem

- Most flows encountered in engineering practice are Turbulent
- Turbulent Flows are characterized by the fluctuating velocity field (in both position and time). We say that the Turbulent velocity field is Random.
- Turbulence highly enhances the rates of mixing of momentum, heat Transfer, mixing of matter etc…

The Problem

- One important aspect of Turbulence is the flow around (bluff) bodies, these include:
  - Flows around buildings
  - Flows around cars
  - Flows around airplanes
  - Flows around airfoils
- A characteristic of such flows is the vortex shedding that occurs in the region behind the body at hand, which produces a periodic force. If this period is close to the natural frequency of the structure, resonance may cause sever damage.
- One well known example is the Tacoma Narrows Bridge that collapsed due to vortex shedding.

The Problem


DISASTER!
The Greatest
Camera Scoop
of all time!
CASTLE FILMS

The Problem

- The primary approach to studying Turbulent flows has been experimental
- However, with the increase of precision & sophistication of eng' applications, experiments (alone) are no more efficient regarding the data extracted from them
- One solution (the only solution we have now!) is to use numerical methods to solve the equations governing a certain flow field

The Problem

Experimental Approaches:

- **Empirical Correlations**
- **Difficulty in reproducing scale models**
- **Difficulty in adjusting flow parameters**
- **High cost**
- **Doubtful accuracy**

Numercial Approaches:

- **Averaging of Equations And Turbulence modeling**
- **Filtering of Equations Large Eddy Simulation**
- **Full resolution of the flow Direct Numerical Simulation**

Equations of Fluid Motion

- Conservation of Mass

$$\frac{\partial \rho}{\partial \tau} + \frac{\partial \rho \tilde{u}_i}{\partial x_i} = 0$$

- Conservation of Momentum

$$\frac{\partial \rho \tilde{u}_i}{\partial \tau} + \frac{\partial \rho \tilde{u}_j \tilde{u}_i}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\mu \frac{\partial \tilde{u}_i}{\partial x_j}\right) - \frac{\partial \tilde{p}}{\partial x_i} + \rho g_i + \tilde{s}_{ui}$$

Equations of Fluid Motion

- Problems of Navier Stokes Equations:
  - Time dependent equations
  - Highly coupled PDEs
  - Infinite number of time & length scales (degrees of freedom contained in the flow)
- Solution: Reduce the number of degrees of freedom from infinity to 1 or 2 by using the Reynolds decomposition

The Reynolds Decomposition

- The Reynolds decomposition is based on the assumption that the turbulent random velocity field is the sum of an average velocity component and a fluctuating component:

$$\tilde{u} = U + u$$

- By doing so, we will be solving the average flow field: a lot of information regarding the small scales are smoothed out.
- For engineering interest this is enough (for the time being!)

Turbulence Models

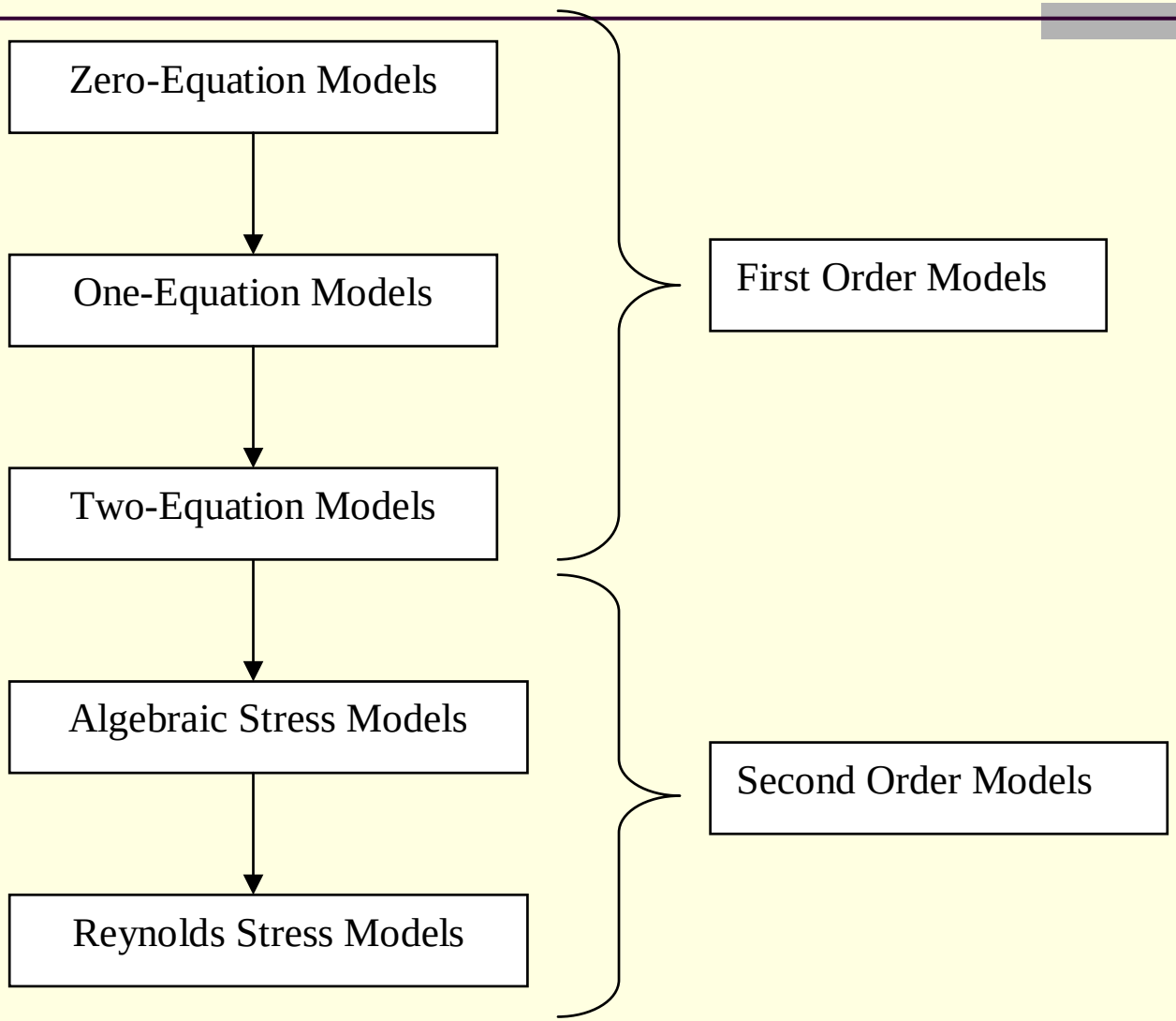
- Using the Reynolds decomposition, the NS equations reduce to:

$$\frac{\partial \rho}{\partial \tau} + \frac{\partial \rho U_i}{\partial x_i} = 0$$

$$\frac{\partial \rho U_i}{\partial \tau} + \frac{\partial \rho U_j U_i}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\mu \frac{\partial U_i}{\partial x_j} - \rho \overline{u_i u_j} \right) - \frac{\partial P}{\partial x_i} + S_{ui}$$

- However, a new problem arises: we have introduced new unknowns: the turbulent stresses.
- The most straightforward solution is to model these new unknowns, which is the basis for turbulence modeling

Turbulence Models



Direct Numerical Simulation

- DNS lies at the upper limit of the numerical solution of the NS equations
- DNS solves the flow field for all time & length scales
- A very fine grid is required which imposes a limit on computational resources!
- DNS is only feasible for low Reynolds number flows which restricts it to the research lab

Direct Numerical Simulation

Re_L	N (# of nodes in each direction)	N^3 (total nodes)	M (time steps)	CPU Time	Time Units
94	104	1.1×10^6	1.2×10^3	20	Min
375	214	1.0×10^7	3.3×10^3	9	H
1,500	498	1.2×10^8	9.2×10^3	13	Days
6,000	1,260	2.0×10^9	2.6×10^4	20	Months
24,000	3,360	3.8×10^{10}	7.4×10^4	90	years
96,000	9,218	7.8×10^{11}	2.1×10^5	5,000	years

Large Eddy Simulation

- An intermediate method between averaging & DNS is Large Eddy Simulation or LES
- In DNS, 96% of the computational effort goes to solving the smallest scales (responsible for dissipating energy) whereas the bulk of energy (practical flow information) is contained in the largest scales comprising 80% of the flow
- Therefore, one might say: why not resolve the large scales of motion exactly & model the small scales
- Doing so is the basic idea behind Large Eddy Simulation

Large Eddy Simulation

- What is the justification behind this thinking? Why can we model the small scales?
- To be able to model the small scales we have to make sure that they are the same for all flows: in fact, this is the case (approximately) and this is proven through the Energy Cascade Mechanism & the underlying kolmogorov Hypotheses

The Energy Cascade

- Turbulence can be considered to be composed of eddies of different sizes
- An eddy is considered to be a turbulent motion localized within a region of size l
- These sizes range from the Flow lengthscale L to the smallest eddies.
- Each eddy of length size ℓ has a characteristic velocity u(l) and timescale t(l)=u(l)/l
- The largest eddies have lengthscales comparable to L

The Energy Cascade

- Each eddy has a Reynolds number
- For large eddies, Re is large, i.e. viscous effects are negligible.
- The idea is that the large eddies are unstable and break up transferring energy to the smaller eddies.
- The smaller eddies undergo the same process and so on
- This energy cascade continues until the Reynolds number is sufficiently small that energy is dissipated by viscous effects: the eddy motion is stable, and molecular viscosity is responsible for dissipation.

Large Eddy Simulation

- In LES, the large scales are directly represented while the small scales are modeled using standard modeling techniques (k-e, RSM…)
- We introduce what is called a filter.
- The filter would act as an automation technique that tells the equations what to fully resolve and what to model.
- The idea is to decompose the velocity field into a filtered field $\bar{U}(x,t)$ and a residual velocity field ***u'(x,t)*** called the residual stress or subgrid scale SGS component.

Large Eddy Simulation

- Filtering is also characterized by what is called a filter width Δ which defines the smallest size of the eddy to be resolved. All eddies with scales less than Δ are modeled.
- Filtering is defined as follows (in one dimension):

$$\bar{\boldsymbol{U}}(\boldsymbol{x},t) = \int_V G(\boldsymbol{r},\boldsymbol{x})\boldsymbol{U}(\boldsymbol{x}-\boldsymbol{r},t)d\boldsymbol{r}$$

Thus, the velocity field has the following decomposition:

$$\boldsymbol{U}(\boldsymbol{x},t) = \bar{\boldsymbol{U}}(\boldsymbol{x},t) + \boldsymbol{u}'(\boldsymbol{x},t)$$

Large Eddy Simulation

Name of Filter	Filter Function
Box	$\frac{1}{\Delta} H(\tfrac{1}{2}\Delta - \lvert r \rvert)$
Gaussian	$\left(\frac{6}{\pi\,\Delta^2}\right)^{1/2} \exp\left(-\frac{6r^2}{\Delta^2}\right)$
Sharp Spectral	$\frac{\sin(\pi r/\Delta)}{\pi r}$
Cauchy	$\frac{a}{\pi\,\Delta\left[\left(r/\Delta\right)^2 + a^2\right]},\ a = \frac{\pi}{24}$

Large Eddy Simulation

- Applying filtering to the original NS equations, we get:

Conservation of Mass:

$$\overline{\left(\frac{\partial U_i}{\partial x_i}\right)} = \frac{\partial \overline{U}_i}{\partial x_i} = 0$$

Conservation of Momentum:

$$\frac{\partial \overline{U}_j}{\partial t} + \frac{\partial \overline{U}_i \overline{U}_j}{\partial x_i} = \nu \frac{\partial^2 \overline{U}_j}{\partial x_i \partial x_i} - \frac{\partial \tau_{ij}^r}{\partial x_i} - \frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_j}$$

By solving the filtered NS equations, we are resolving the large scales (all eddies Of sizes>Δ) while we are modeling the small scales (all eddies of sizes<Δ). However, new unknowns are introduced in the stress tensor, called the Subgrid Scale Stress Tensor (SGS) which requires modeling.

The Smagorinsky Model

- The SGS tensor is responsible for the transfer of energy from the large scales to the small scales. Modeling the small scales therefore means modeling of the SGS tensor. The standard model is called the Smagorinsky model & is an eddy viscosity model based on the mixing length hypothesis.

$$\tau_{ij}^{r} = -2\nu_r \overline{S}_{ij} \qquad \overline{S}_{ij} = \frac{1}{2}\left(\frac{\partial \overline{U}_i}{\partial x_j} + \frac{\partial \overline{U}_j}{\partial x_i}\right)$$

Filtered Rate of Strain

$$\nu_r = \ell_s{}^2 \overline{\mathsf{S}} = \ell_s{}^2 \sqrt{2\overline{S}_{ij}\overline{S}_{ij}}$$

Eddy viscosity of the residual motions. Modeled using mixing length theory.

$$\ell_s = C_s \Delta$$

Smagorinsky length scale, proportional to the filter width.

The Smagorinsky Model

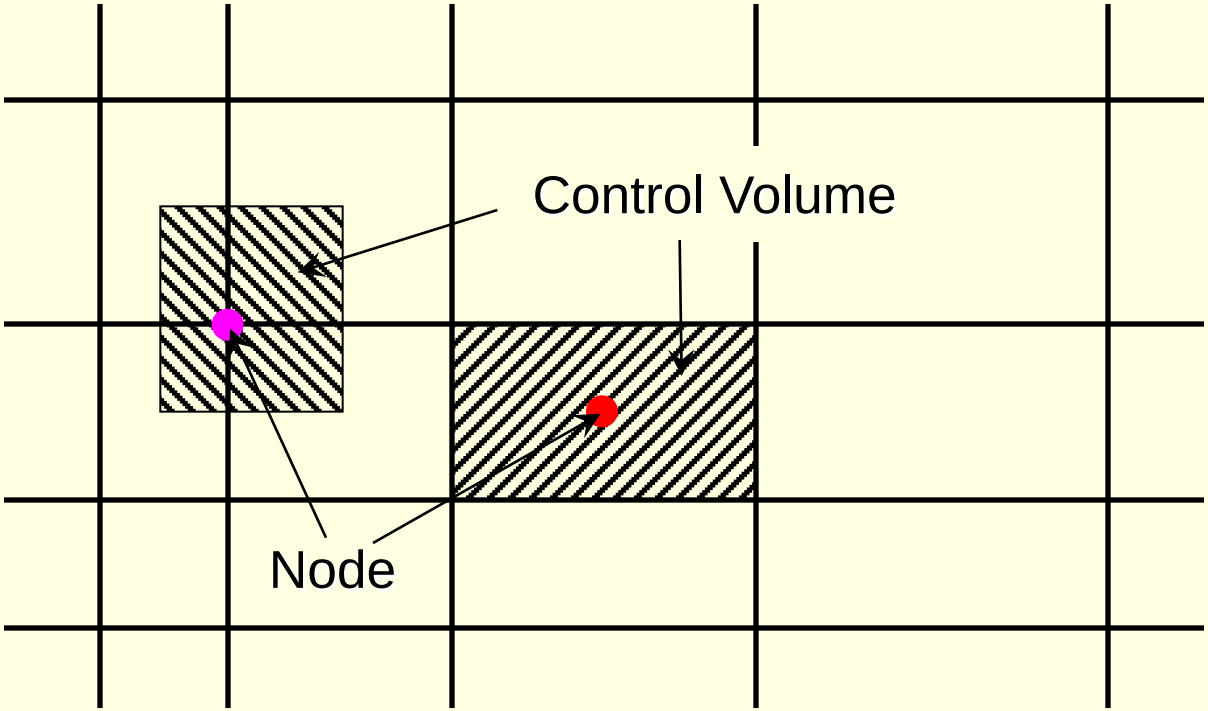
- Modeling the SGS tensor reduces now to modeling the lengthscale $\ell_s = C_s \Delta$
- Cs is the Smagorinsky constant & has a value that lies between 0.12 & 0.2
- Δ is the filter width & is usually taken as:
  - $\Delta = (dx \cdot dy \cdot dz)^{1/3}$ or $\Delta = (dx^2 + dy^2 + dz^2)^{1/2}$
  - We have used both approaches in our code
- However, this model is not without problems: In shear flows, the smagorinsky constant has to be reduced near wall regions to around 0.06 which is solved by using dynamic models

Numerical Methods

- Finite Volume Methods
- Spatial Discretization: The power law scheme
- Time Discretization: The Euler implicit method

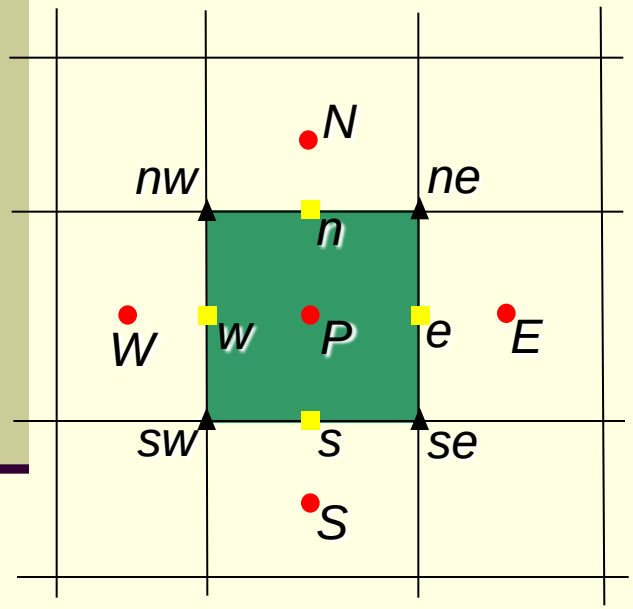
Finite Volume Methods

- Finite Volume Methods:
  - The domain is divided into a finite number of computational cells known as control volumes



Finite Volume Methods

- Surface & volume integrals are approximated over each control volume as follows:



$$F_e = \int_{\Delta Se} f dS = \overline{f}_e \Delta S_e \approx f_e \Delta S_e$$

$$F_e = \int_{\Delta Se} f dS = \overline{f}_e \Delta S_e \approx \frac{f_{ne} + f_{se}}{2} \Delta S_e$$

$$F_e = \int_{\Delta Se} f dS = \overline{f}_e \Delta S_e \approx \frac{f_{ne} + f_{se} + 4 f_e}{6} \Delta S_e$$

$$\int_{\forall} S_\Phi dc = \overline{S}_\Phi \Delta \forall \approx S_{\Phi P} \Delta \forall$$

Spatial Discretization

- The power law scheme has been adopted in our project to discretize space.
- The main advantage of this scheme is its superior stability. It is a first order scheme.

$$\psi_E = \begin{cases} \psi_E & if \quad Pe_e \le -10 \\ \psi_E + (\psi_E - \psi_P)\dfrac{1-(1-0.1|Pe_e|)^5}{Pe_e} & if \quad -10 \prec Pe_e \prec 0 \\ \psi_p + (\psi_E - \psi_P)\dfrac{1-(1-0.1|Pe_e|)^5}{Pe_e} & if \quad 0 \prec Pe_e \prec 10 \\ \psi_p & if \quad 10 \le Pe_e \end{cases}$$

Time Discretization

- For unsteady flows, like in our project, we have to discretize the unsteady term (term in function of time).
- There are two methods: either explicit or implicit.
- We have used an implicit method because of implicit methods have superior stability & the ability to adapt to generally any time step considered
- The method used is called the Euler Implicit Method
- Depending on the finite difference scheme used we end up with either a first order or second order result.
- We have used second order approximations based on the Central difference schemes.

Time Discretization

- Transport equation in One dimension

$$\frac{\partial \phi}{\partial t} = -u\frac{\partial \phi}{\partial x} + \frac{\Gamma}{\rho}\frac{\partial^2 \phi}{\partial x^2}$$

- After discretization using CDS, we get:

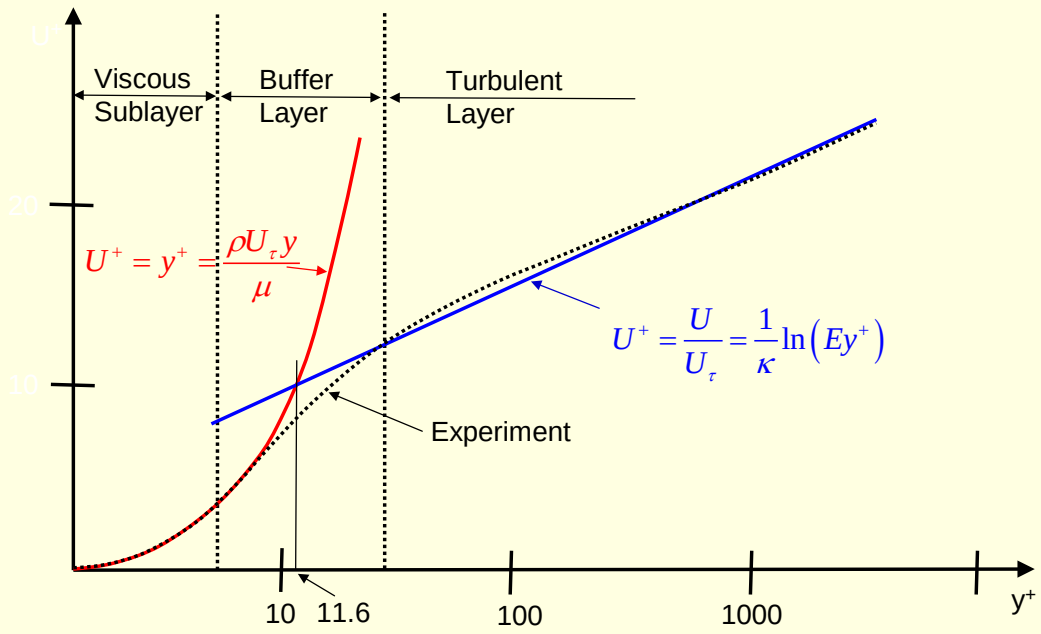
$$\phi_i^{n+1} = \phi_i^n + \left[-u\frac{\phi_{i+1}^{n+1} - \phi_{i-1}^{n+1}}{2\Delta x} + \frac{\Gamma}{\rho}\frac{\phi_{i+1}^{n+1} + \phi_{i-1}^{n+1} - 2\phi_i^{n+1}}{(\Delta x)^2} \right]\Delta t$$

Boundary Conditions

- Wall Boundary Condition
- OutFlow Boundary Condition
- InFlow (Inlet) Boundary Condition

Wall Boundary Condition

- Near a wall, the flow has a complex structure
- The no slip boundary condition is used in our project
- The law of the wall is used to approximate the velocity



OutFlow Boundary Condition

- For OutFlow BC, we specify a zero gradient (flux) of either pressure or velocity.
- In our project, we have specified a zero gradient of velocity at the outlet
- The ouflow BC should be located in a region where the flow is fully developed & far away from the inlet

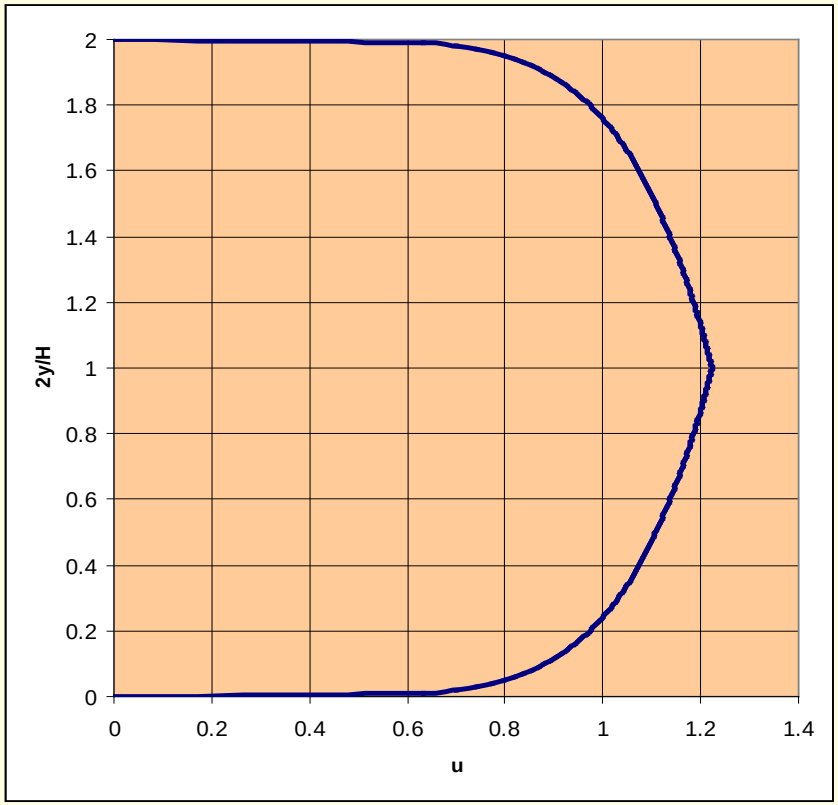
InFlow Boundary Condition

- At the inlet, we specify a velocity profile, either uniform or parabolic
- In order to correctly simulate a turbulent flow, we have to introduce a fully developed turbulent velocity inlet, that’s why we have chosen a parabolic inlet with a boundary layer thickness of half the channel height
- If we were to choose a uniform velocity profile, we would have to position to body at a region far enough from the inlet for the flow to become fully developed

InFlow Boundary Condition

- Parabolic Inlet

$$u = 1.224 U_{mean} \left(\frac{2y}{H} \right)^{1/7}$$



InFlow Boundary Condition

- However, for LES, since the filtered equations are somehow deterministic & tend to smooth out all turbulent fluctuations, we are obliged to impose artificial random fluctuations on the velocity inlet
- This is done by jittering the inlet parabolic profile with random number of 2% of the mean value of velocity

InFlow Boundary Condition

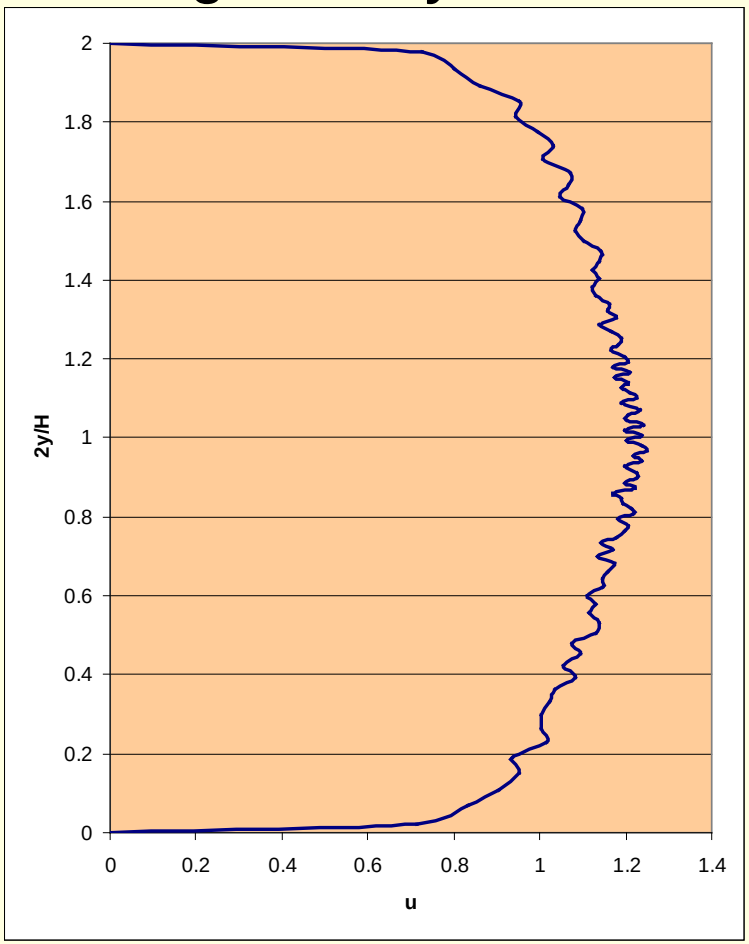
- Generation of random numbers:
  - First generate a set of random numbers obeying the normal distribution in our project such that *0<r<1*
  - Now generate a integer set of random numbers such that: *i = 10^6.r &* round it off
  - Generate the final set of random numbers obeying:

$$R = (-1)^i r \cdot p$$

Where P is the percentage

InFlow Boundary Condition

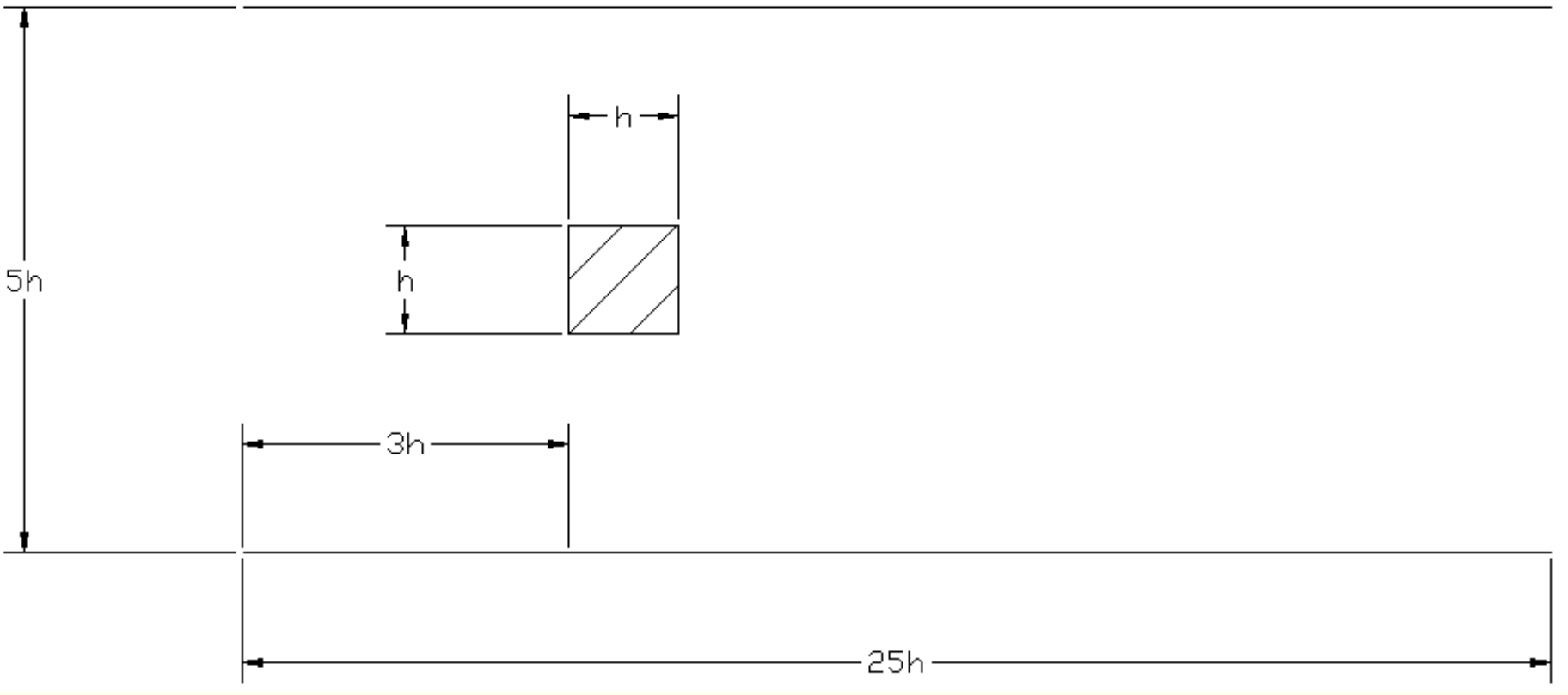
- The final fluctuating velocity inlet becomes:



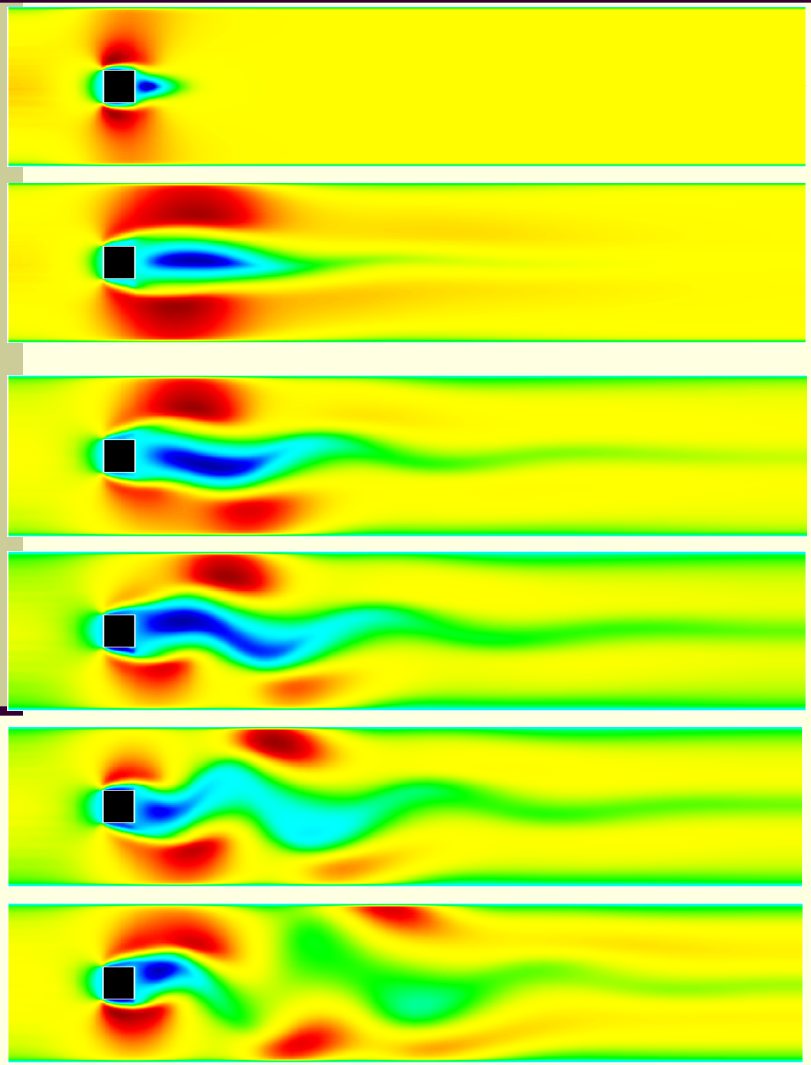
Code Validation

- Geometry
- Qalitative Results
- Vortex Shedding
- Quantitative Results

Geometry

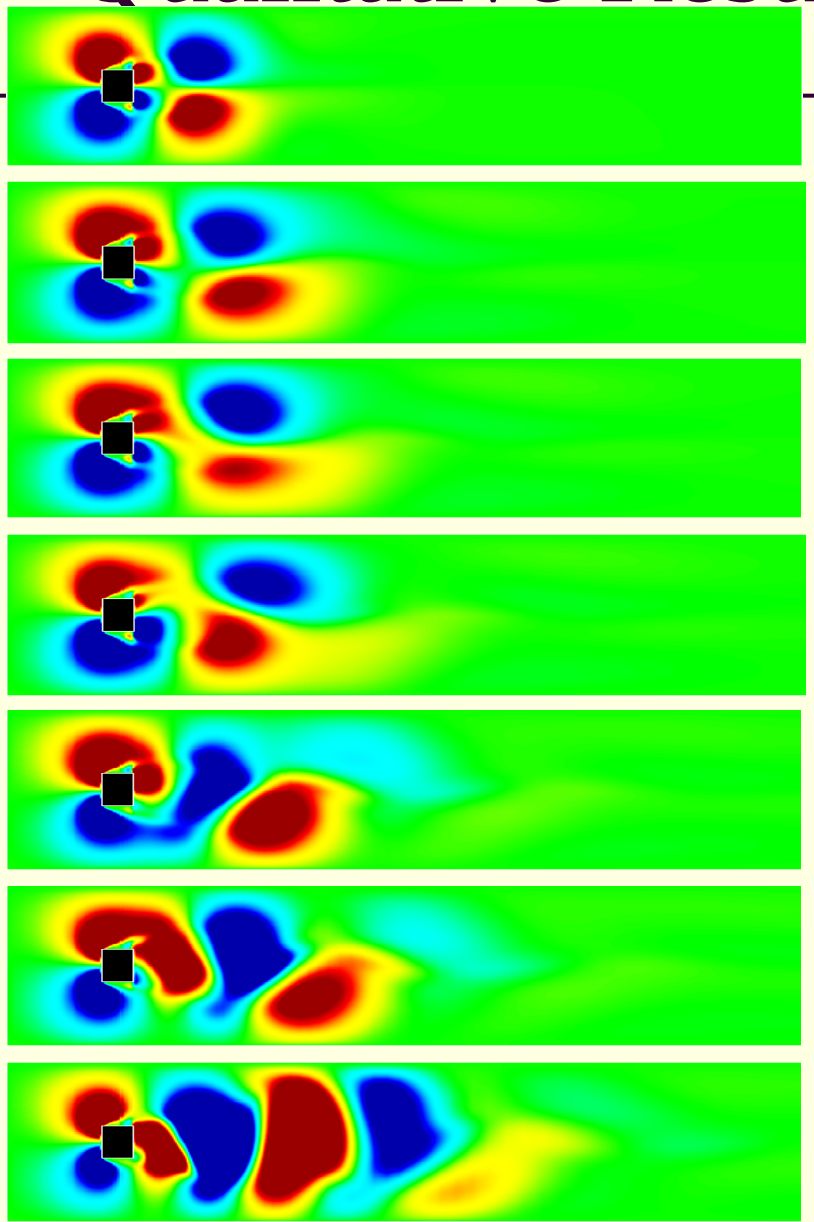


Qualitative Results

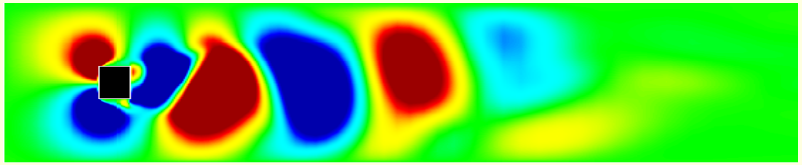


Contours of Streamwise velocity at various time steps

Qualitative Results



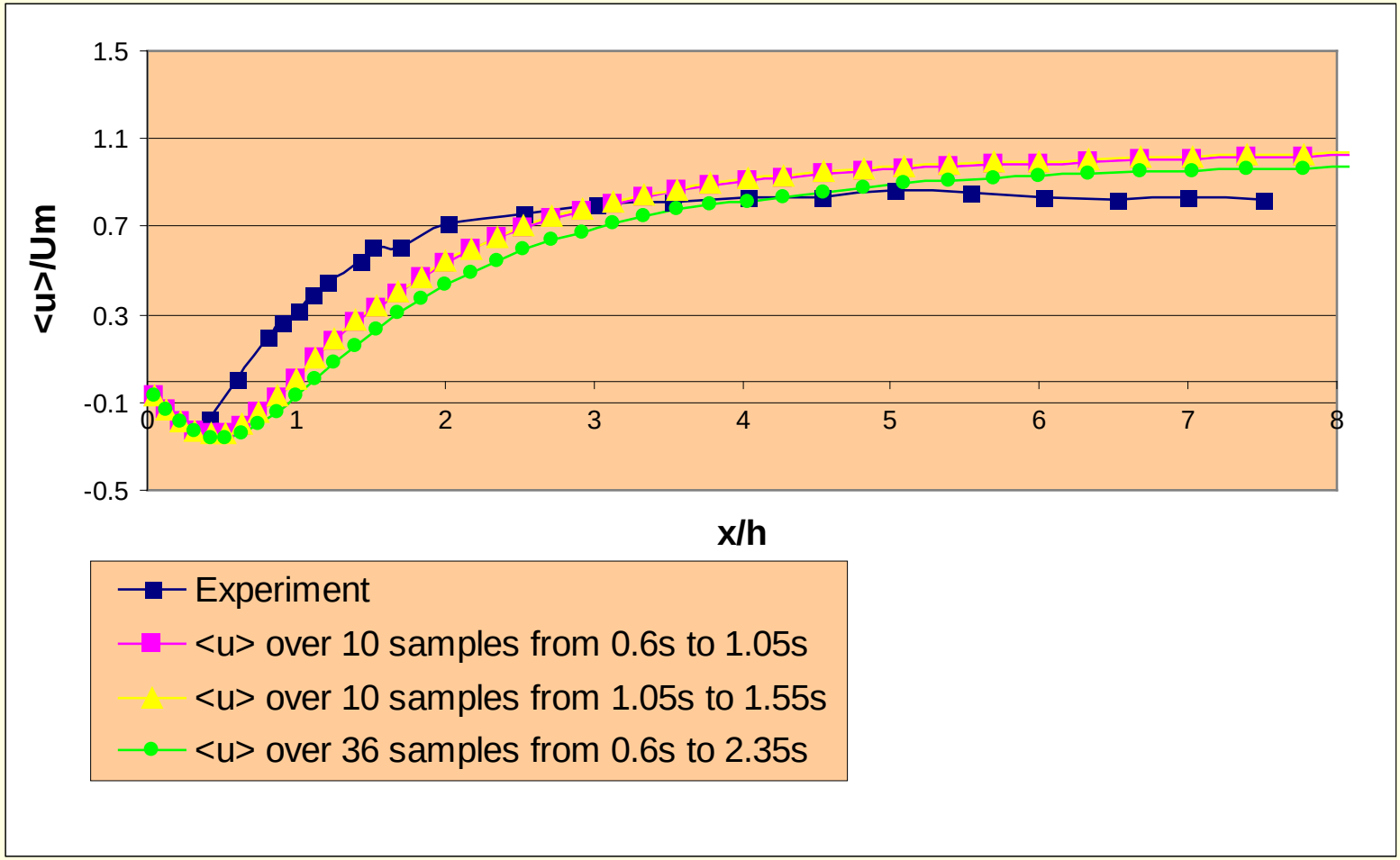
Contours of Normal velocity at various timesteps



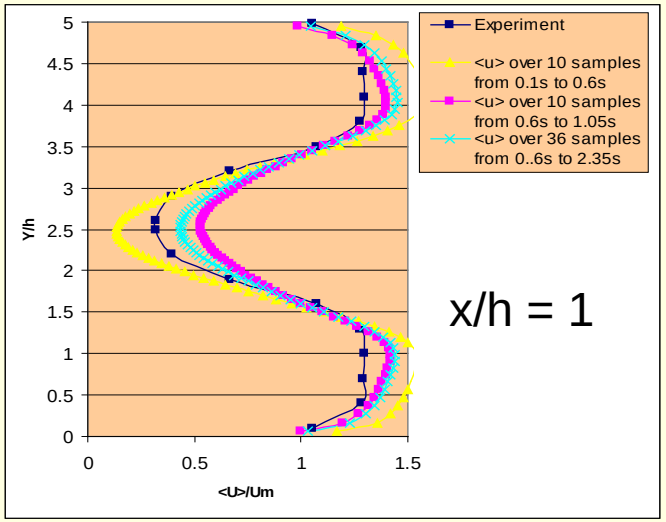
Vortex Shedding

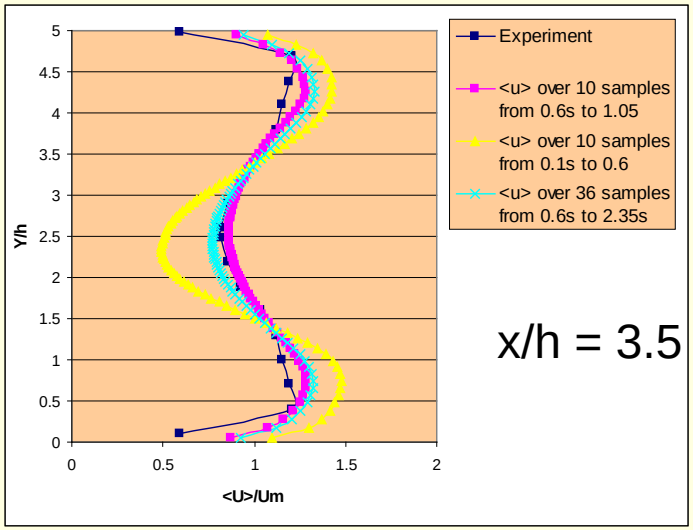
- Vortex Shedding: Streamwise Velocity
- Vortex Shedding: Normal Velocity

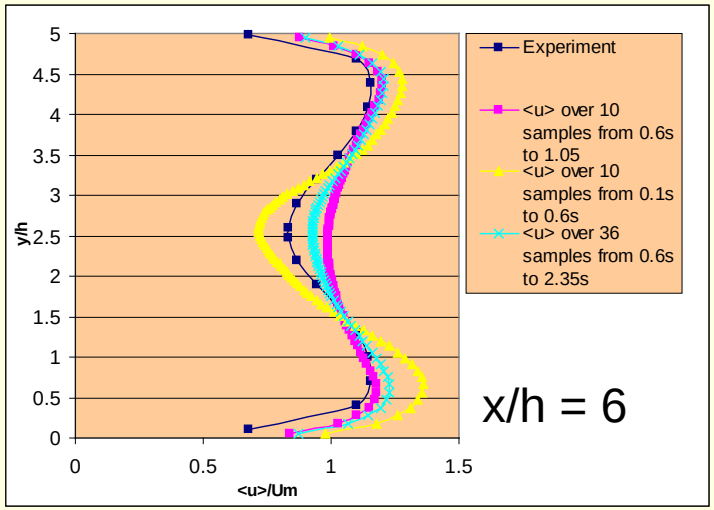
Centerline Velocity

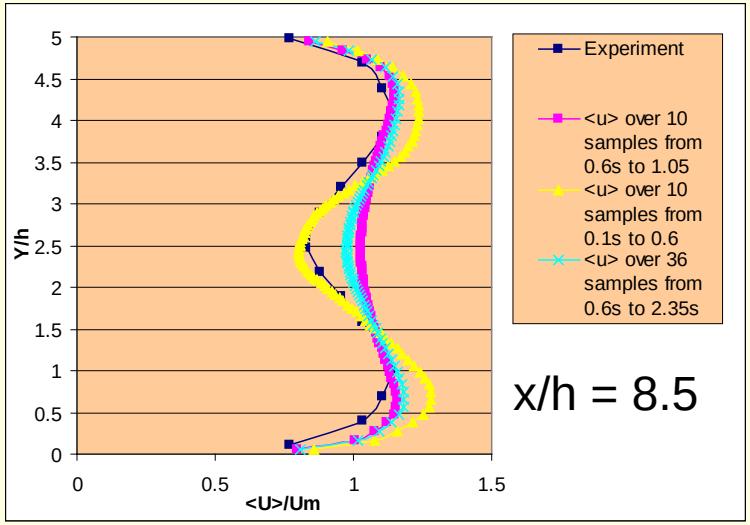


Streamwise Velocity

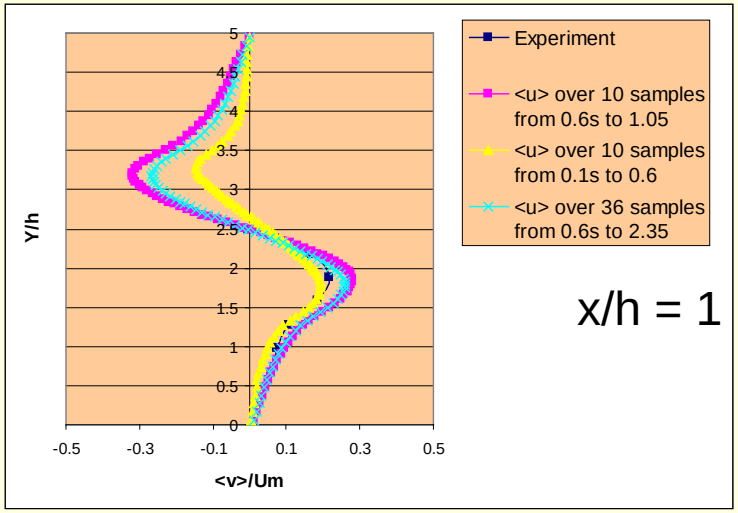


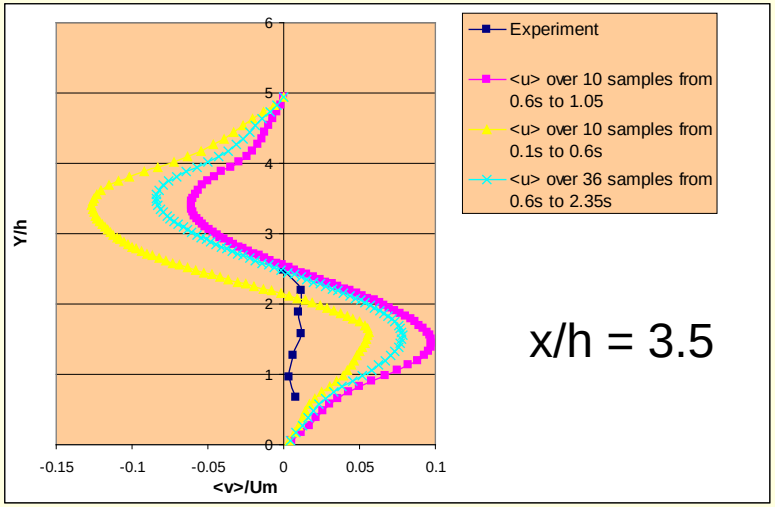


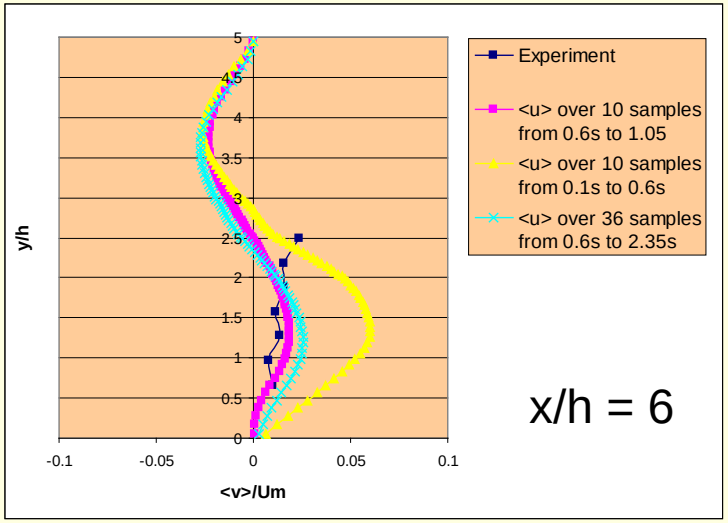


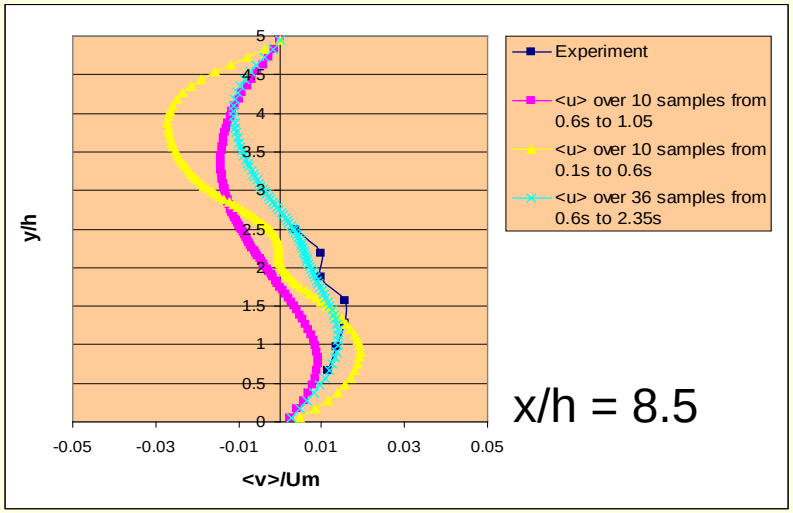


Normal Velocity

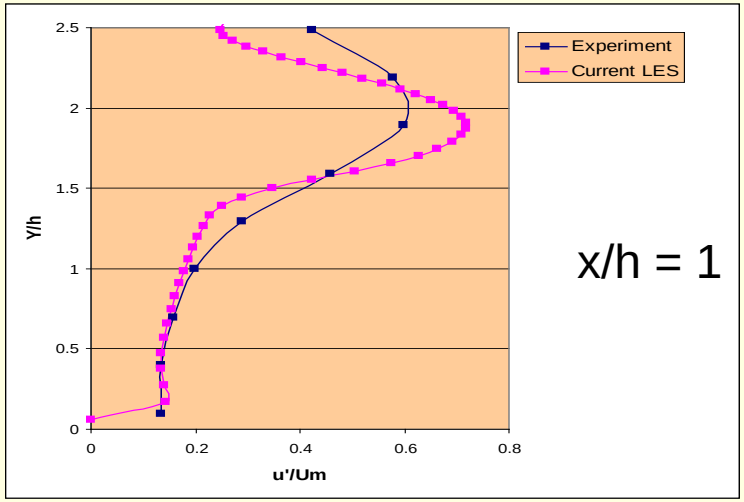


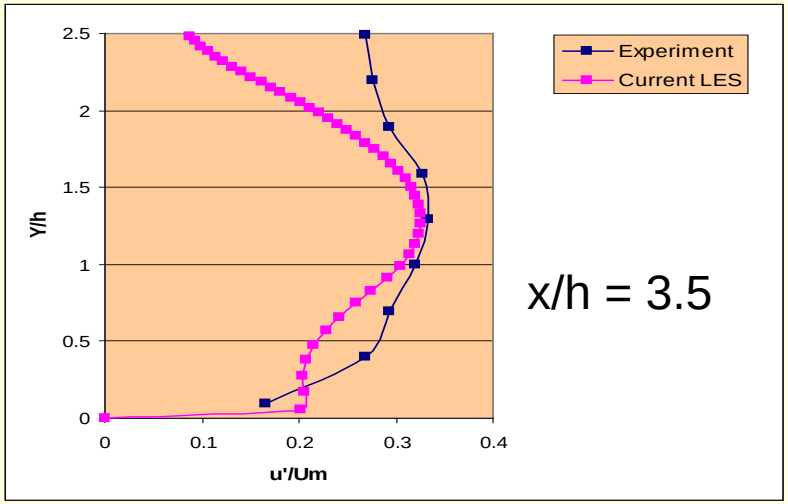


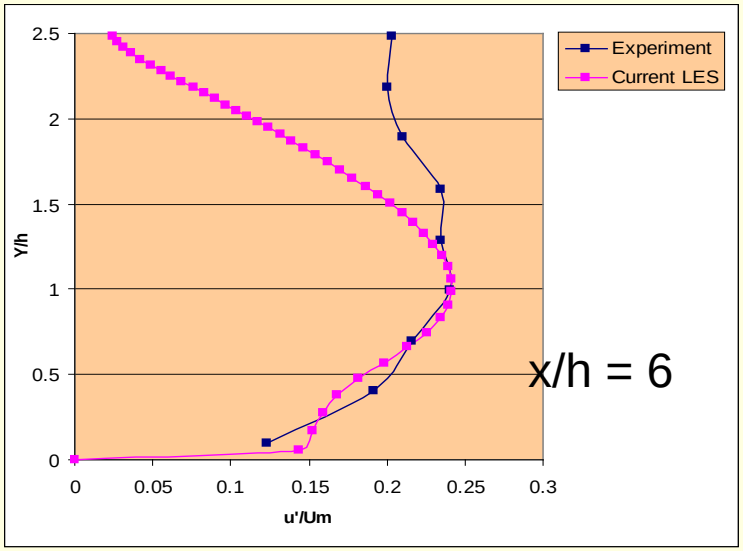


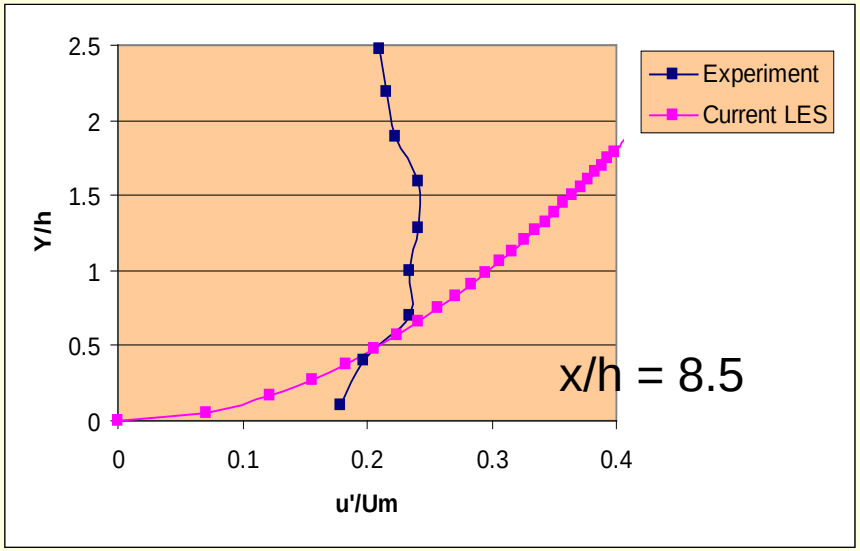


Reynolds Normal Stress

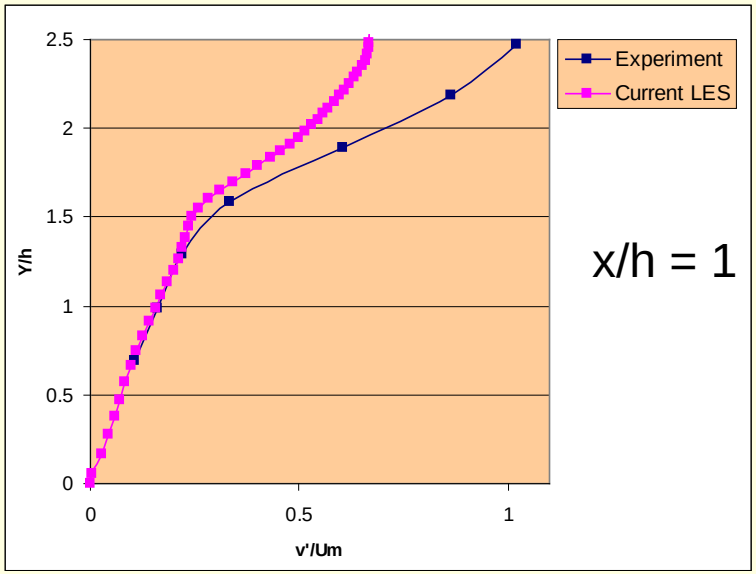


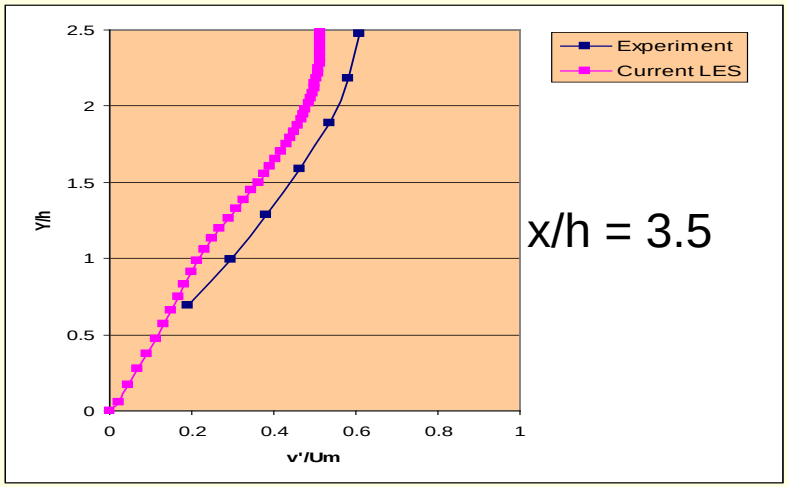


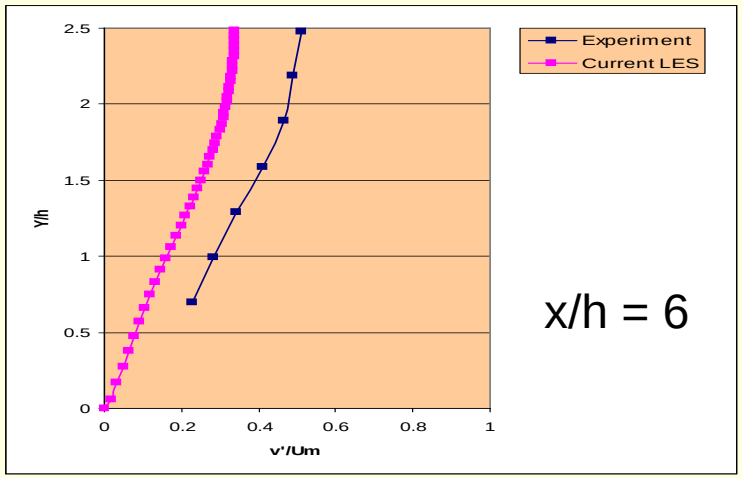


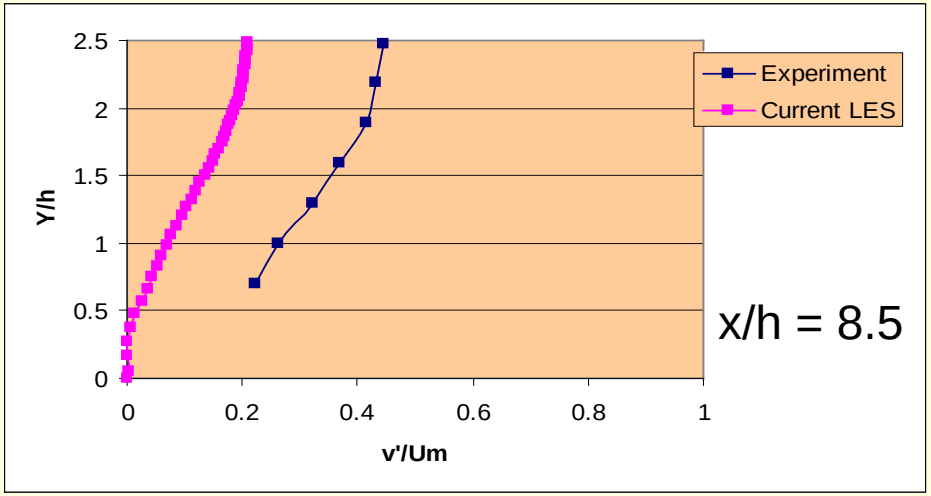


Reynolds Normal Stress

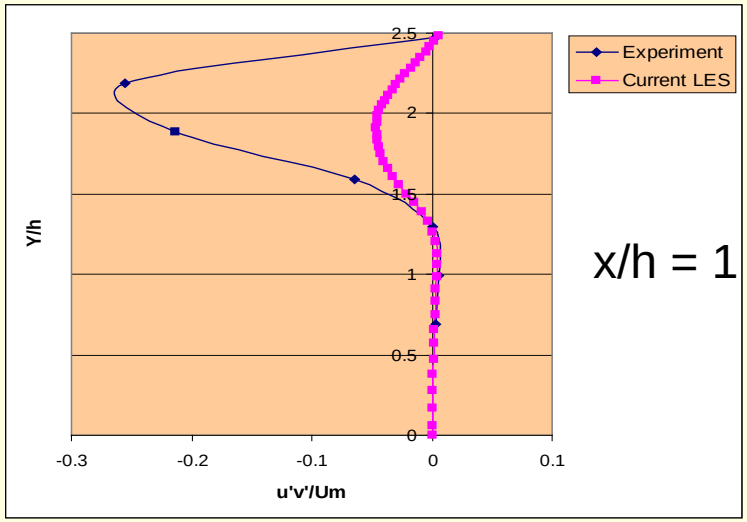


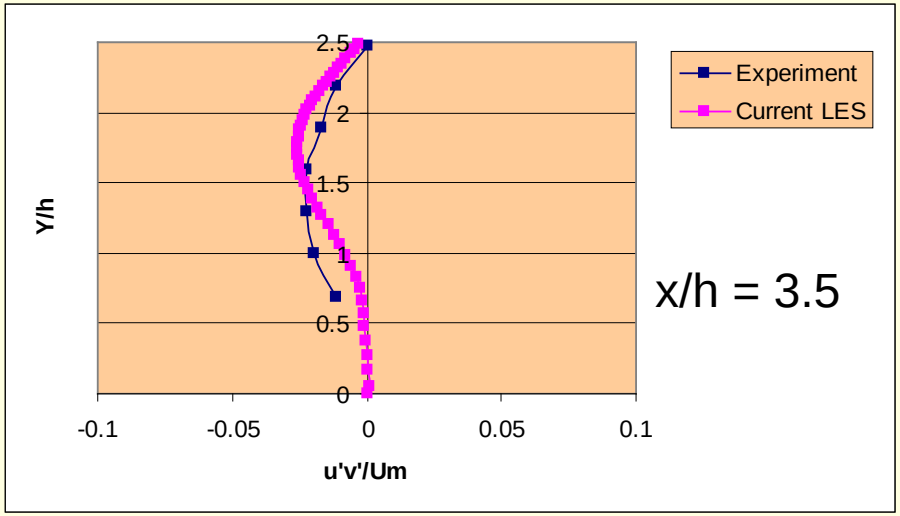


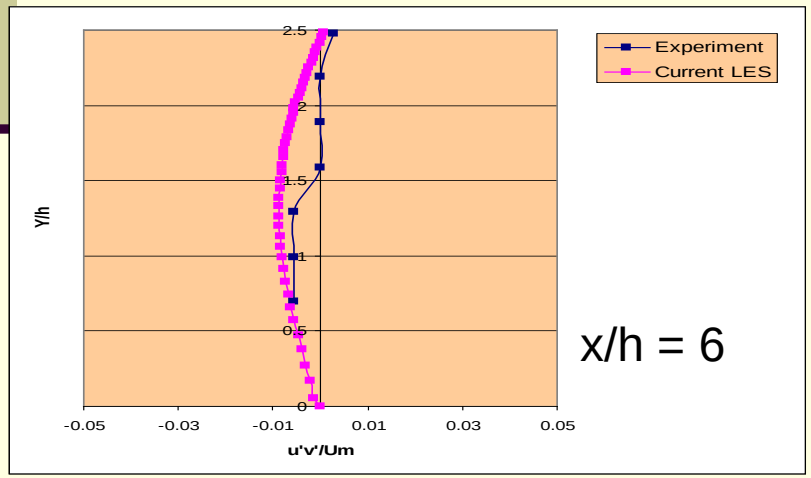


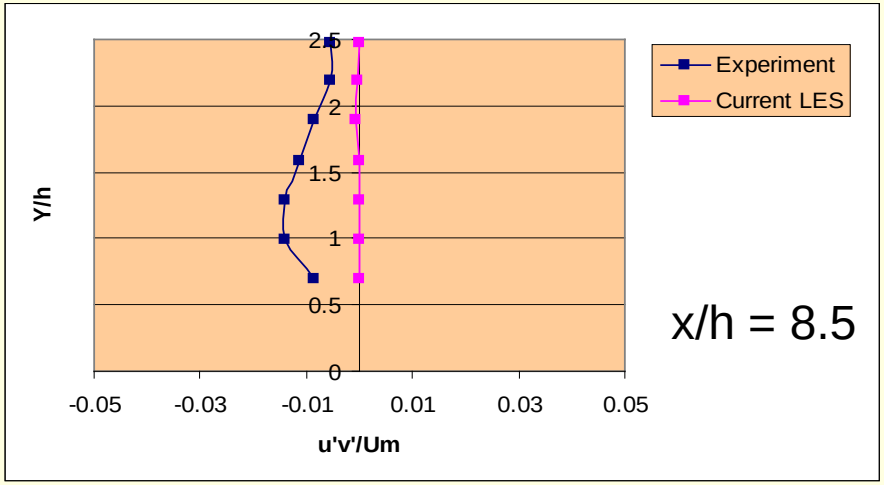


Reynolds Shear Stress









Thank You!