# SIMULATION OF TURBULENT FLOW OVER A BACKWARD FACING STEP USING LATTICE BOLTZMANN METHOD

NOR AZWADI C. SIDIK*, JAVAD GHADERIAN, SAKINEH SADEGHIPOUR

Faculty of Mechanical Engineering, Universiti Teknologi Malaysia,
81310 UTM Skudai, Skudai, Johor DT, Malaysia
*Corresponding Author: azwadi@fkm.utm.my

**Abstract**

The lattice Boltzmann scheme for solving the equations governing the flow of fluids has become increasingly popular in recent years. In this method, a probability distribution function for the velocity of fluid elements is computed at each grid point, which contrasts with more established computational fluid dynamics methods, which solve for an average velocity. In this paper the effectiveness of the lattice Boltzmann equation (LBE) as a computational tool for performing large-eddy simulations (LES) of turbulent flows was assessed. Three-dimensional turbulent flow over backward facing step was considered for this investigation. LBE-LES and Navier Stokes Direct Numerical Simulation (NS-DNS) results for this case has been compared and it was observed that the LBE-LES simulation is in good agreement with other computational simulations.

Keywords: Lattice Boltzmann method, Backward step, Turbulent flow, Large eddy simulation.

## 1. Introduction

In recent years, Lattice Boltzmann Method (LBM) has been known as an alternative numerical scheme for simulating many types of fluid flow and heat transfer to traditional methods [1-3]. The LBM is based on the Boltzmann equation, which governs the dynamics of molecular distribution function in microscopic view. In the LBM, the Boltzmann equation discrete by a specific lattice and distribution functions would be obtained by iterating numerical method in that lattice. The macroscopic variables are obtained from distribution functions. The resulting formulation can yield computational advantages, easy implementation of boundaries and fully parallel algorithms over traditional continuum

**Nomenclatures**

| | |
|---|---|
| $c_s$ | Speed of sound, m/s |
| $c_{sm}$ | Kolmogorov constant |
| $e_\alpha$ | Microscopic velocity, m/s |
| $f_\alpha$ | Distribution function |
| $f_\alpha^{eq}$ | Equilibrium distribution function |
| $T$ | Time, s |
| $U$ | Flow velocity, m/s |

*Greek Symbols*

| | |
|---|---|
| $\mu$ | Dynamic viscosity, kg/m.s |
| $\upsilon$ | Kinematic viscosity, $m^2$/s |
| $\rho$ | Density, kg/$m^3$ |
| $\tau$ | Time relaxation, s |
| $\omega_\alpha$ | Weighting factor |

methods. In particular, the LBM method has been successfully applied to problems which are usually difficult for traditional numerical schemes, such as fluid flows through porous media, multiphase fluid flows and suspension motions in fluids [1, 4-6]. On the other hand, the most practical flows are in turbulent regime. Therefore the simulation of turbulent flow using the Lattice Boltzmann scheme can be useful.

Turbulent flows are characterized by the occurrence of eddies, whose sizes may vary over a large range. Larger eddies contain the main portion of the flow's energy, which is successively transferred to smaller eddies, and which is eliminated by viscous dissipation in the smallest eddies. This process is described by the theory of Kolmogorov [7], which predicts that the size of the smallest eddies is proportional to, $\eta \sim Re^{-3/4}$ where Re is the Reynolds number. In the numerical simulation of a turbulent flow, the smallest eddies must be resolved by the numerical grid. Given three space dimensions, this requires N~O($Re^{9/4}$) grid points in the discretization. The method that involves the discretization of the grid sufficiently fine for resolving all occurring eddies is known as direct numerical simulation (DNS). In industrial applications such as aerodynamic investigations of automobiles, typical Reynolds numbers lie at $10^6$ and above. Hence, to properly solve these types of problems, one would require over $10^3$ grid points, a number which is inaccessible in terms of storage space or CPU performance, even by large parallel computers. In a direct numerical simulation, all eddies down to the smallest are resolved by the grid. To achieve better performance, turbulence model avoids to simulate the small scales and replaces their effect by appropriate ''subgrid'' models. This kind of turbulence modeling which is called "Large Eddy Simulation" (LES) technique has been first incorporated into the Lattice Boltzmann (LB) framework during the past decade [8-9]. In the recent years, the combination of LES and LBE has led to good results [4, 5, 10]. Because LES involves modeling of the smaller eddies, the smallest finite-difference cells can be much larger than the Kolmogorov length, and much larger time steps can be taken than those used in a DNS technique. Hence, for a given computing cost, it is possible to achieve much higher Reynolds numbers with LES than with DNS. We

implemented LES in the lattice Boltzmann frame to simulate the three-dimensional turbulent flow over a backward-facing step.

Backward facing step is chosen for its apparent geometrical simplicity, but it involves relatively complex flow phenomena. This geometry is well suited to study the turbulence behavior under separation, recirculation and reattachment phenomena [11]. Different flow parameters, reattachment length, velocity field, velocity profiles and turbulent properties were computed and compared with DNS data set [12].

## 2. Computational Method

The LBM with single-relaxation-time approximation due to Bhatnagar, Gross, Krook (BGK) model for the collision operator (this collision operator also called Single Relaxation Time) is [13-15].

$$f_\alpha(\mathbf{x} + \mathbf{e}_\alpha \delta t, t + \delta t) = f_\alpha(\mathbf{x}, t) - \frac{1}{\tau}\left(f_\alpha - f_\alpha^{eq}\right) \tag{1}$$

where $f_\alpha$ is the density distribution function with discrete velocity $\mathbf{e}_\alpha$ along the $\alpha^{\text{th}}$ direction, $f_\alpha^{eq}$ is the equilibrium distribution function, and $\tau$ is the relaxation time due to fluid particle collisions. Eq. (1), termed the Lattice Boltzmann Equation (LBE) with BGK approximation or LBGK model, is usually solved in the following two steps:

Collision step:

$$\widetilde{f}_\alpha(\mathbf{x}, t) = f_\alpha(\mathbf{x}, t) - \frac{1}{\tau}\left(f_\alpha - f_\alpha^{eq}\right) \tag{2}$$

Streaming step:

$$f_\alpha(\mathbf{x} + \mathbf{e}_\alpha \delta t, t + \delta t) = \widetilde{f}_\alpha(\mathbf{x}, t) \tag{3}$$

where $\widetilde{f}_\alpha$ represents the post-collision state. The collision time-scale determines the viscosity $\mu$ of the modeled fluid. In what follows, we use the LBE model with nine velocities in two dimensions, i.e., the D2Q9 model.

The equilibrium distribution functions for incompressible flow are [6]

$$f_\alpha^{eq} = \omega_\alpha\left[\delta\rho + \rho_0\left(3\mathbf{e}_\alpha \cdot \mathbf{u} + 4.5(\mathbf{e}_\alpha \cdot \mathbf{u})^2 - 1.5\mathbf{u}^2\right)\right] \tag{4}$$

where $\delta\rho$ is the density fluctuation, $\rho_0$ is the constant mean density in the system which is usually set to 1. The total density is $\delta\rho + \rho_0$. The practice of using only $\delta\rho$ instead of $\rho$ in Eq. (4) is to reduce the effect due to the round-off error in the LBE simulations [16-17]. The weight factors $\omega_\alpha$ for the D2Q9 model are:

$$\omega_\alpha = \begin{cases} 4/9, \alpha = 0 \\ 1/9, \alpha = 1,3,5,7 \\ 4/9, \alpha = 2,4,6,8 \end{cases} \tag{5}$$

The mass and momentum conservations are strictly enforced by

$$\rho = \sum_{\alpha=0}^{8} f_\alpha$$
$$\rho\mathbf{u} = \sum_{\alpha=0}^{8} \mathbf{e}_\alpha f_\alpha$$
(6)

The kinematic viscosity has the following relation with the relaxation time

$$\upsilon = \left(\tau - \frac{1}{2}\right) c_s^2 \delta t$$
(7)

where $c_s$ is the lattice sound speed.

## Large eddy simulation (LES)

Large eddy simulation (LES) is a popular technique for simulating turbulent flows [4, 5, 9, 10]. An implication of Kolmogorov's theory of self similarity is that the large eddies of the flow are dependent on the geometry while the smaller scales are more universal. This feature allows one to explicitly solve for the large eddies in a calculation and implicitly account for the small eddies by using a sub grid-scale model (SGS model).

Mathematically, one may think of separating the velocity field into a resolved and sub-grid part. The resolved part of the field represent the "large" eddies, while the subgrid part of the velocity represent the "small scales" whose effect on the resolved field is included through the sub grid-scale model. To understand the primary difference between DNS and LES, we introduce the concept of filtering. The filter function separates a variable in subgrid and resolved scale.

$$\overline{u}_i(\mathbf{x}) = \int G(\mathbf{x} - \varsigma) u(\mathbf{x}) d\zeta$$
(8)

Resulting in

$$u_i = \overline{u}_i + u_i'$$
(9)

where $\overline{u}_i$ is the resolvable scale part and $u_i'$ is the sub grid-scale part. However, most practical (and commercial) implementations of LES use the grid itself as the filter (the box filter) and perform no explicit filtering. If we filter the Navier-Stokes (NS) equations, it results in

$$\frac{\partial \overline{u}_i}{\partial t} + \overline{u}_j \frac{\partial \overline{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}\left((\upsilon + \upsilon_t)\frac{\partial \overline{u}_i}{\partial x_j}\right)$$
(10)

where $u_i$ is the $i^{th}$ component of velocity vector and $\upsilon_t$ is the sub grid-scale turbulent viscosity. The filtered form of NS is similar to ordinary NS, but the hydrodynamics variables replaced with filtered form, and a turbulent viscosity is added to the molecular viscosity.

In the Smagorinsky-Lilly model, the eddy viscosity is modeled by

$$\mu_t = (c_s \Delta)^2 \overline{s}$$
(11)

where the filter width is usually taken to be

$$\Delta = \left(\text{volume}_{\text{grid}}\right)^{1/3} \tag{12}$$

$$\bar{s} = \sqrt{2s_{ij}s_{ij}} \tag{13}$$

The effective viscosity is calculated from

$$\mu_{\text{total}} = \mu_t + \mu_0 \tag{14}$$

The Smagorinsky constant usually has the value as

$$c_{sm} = [0.1, 0.2] \tag{15}$$

The fundamental question here is how to modify the lattice Boltzmann method discussed in the previous section to simulate the filtered density. In order to apply the subgrid idea in the framework of LB method, let us introduce the filtered particle distribution, defined as follows

$$\bar{f}_\alpha(\mathbf{x}) = \int f_\alpha(\mathbf{x}) G(\mathbf{x}, \mathbf{x}') d\mathbf{x}' \tag{16}$$

Now we filter the Eq. (1).

$$\int \left[ f_\alpha(\mathbf{x} + \mathbf{e}_\alpha \delta t, t + \delta t) - f_\alpha(\mathbf{x}, t) = \frac{1}{\tau}\left(f_\alpha - f_\alpha^{eq}\right) \right] G(\mathbf{x}, \mathbf{x}') d\mathbf{x}' \tag{17}$$

Filtering the left side of Eq. (17) is straight forward, but when filtering is applied on the collision term, or the equilibrium distribution function on the right side of the Eq. (17), several correlation terms are obtained due to the nonlinearity of velocity in the equilibrium distribution function. A second and more direct approach is to link the collision steps with some local information and to abandon the single-time-relaxation approximation. To do this, we assume that the filtered particle distribution will approach a local filtered equilibrium distribution, which can be chosen to depend only on the local filtered mean quantities, and the forms of the equilibrium distribution function are all the same as in Eq. (2), except that we will now use the filtered quantities to replace the unfiltered quantities. It means, instead of calculating

$$\bar{f}_\alpha^{eq}(\mathbf{x}) = \int f_\alpha^{eq}(\mathbf{x}) G(\mathbf{x}, \mathbf{x}') d\mathbf{x}' \tag{18}$$

From Eq. (4) we assume

$$\bar{f}_\alpha^{eq}(\mathbf{x}) = \omega_\alpha \left[ \delta\bar{\rho} + \bar{\rho}_0 \left( 3\mathbf{e}_\alpha \cdot \bar{\mathbf{u}} + 4.5(\mathbf{e}_\alpha \cdot \bar{\mathbf{u}})^2 - 1.5\bar{\mathbf{u}}^2 \right) \right] \tag{19}$$

As it was shown in Eq. (10), by filtering NS equations and using the filtered form of hydrodynamic quantities, the eddy viscosity is produced. Here, using the filtered form of hydrodynamic quantities in equilibrium distribution function, this eddy viscosity is produced again. This eddy viscosity acted on the relaxation time, and the turbulent relaxation time is produced or the effect of the correlation will only locally introduce an eddy viscosity and the relaxation time is allowed to depend on space. Explicitly, we will incorporate the Smagorinsky model for eddy viscosity into the relaxation time.

In the LES an extra eddy viscosity and turbulent relaxation time is introduced

$$\tau_{\text{total}} = 3\upsilon_{\text{total}} + \frac{1}{2} \tag{20}$$

Defining

$$\tau_{\text{total}} = \tau_0 + \tau_t \tag{21}$$

this yields

$$\tau_{\text{total}} = 3(\upsilon_0 + \upsilon_t) + \frac{1}{2} = 3\left(\upsilon_0 + \frac{1}{2}\right) + 3\upsilon_t \tag{22}$$

From Smagorinsky model, we have

$$\upsilon_t = (c_{sm}\Delta)^2 \bar{s}$$
$$\tau_t = 3(c_{sm}\Delta)^2 \bar{s} \tag{23}$$

So the filtered form of the LBE-LES method is modified as

$$\bar{f}_\alpha(\mathbf{x} + \mathbf{e}_\alpha \delta t, t + \delta t) = \bar{f}_\alpha(\mathbf{x}, t) - \frac{1}{\tau_0 + \tau_t}\left(\bar{f}_\alpha - \bar{f}_\alpha^{eq}\right) \tag{24}$$

## 3. Initialization of Simulation

Figure 1 shows the schematic view of the flow domain used in the three-dimensional simulation. The computational domain consists of a streamwise length $L_x = 30h$, including an inlet section $L_i = 10h$ prior to the sudden expansion, vertical height $L_y = 6h$ and spanwise width $L_z = 4h$, where $h$ is the step height. The coordinate system is placed at the lower step corner as shown in Fig. 1. The Reynolds number based on step-height is defined as Re= $\rho Uh/\mu$, where $U$ is mean inlet velocity. A uniform computational grids $(N_x \times N_y \times N_z) = (300 \times 60 \times 40)$ was used. This grid size has chosen to capture the minimum eddy size which is needed for LES. Using a coarser gird can reduce the accuracy of results. The expansion ratio is $ER = L_y/(L_y - h) = 1.2$.
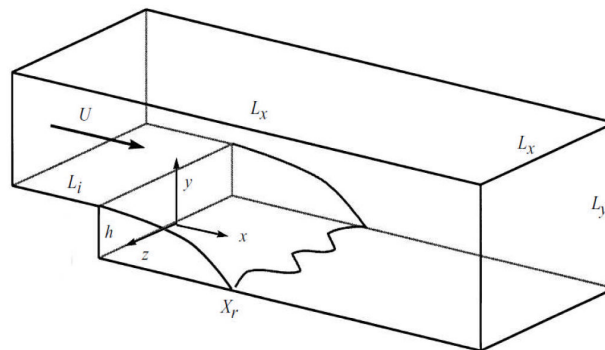


**Fig. 1. Backward Facing Step Flow Configuration [11].**

The boundary condition is chosen to be consistent with DNS by Le et al. [12]. In the spanwise direction, the flow is assumed to be statistically homogeneous

and periodic boundary conditions are used. No-slip boundary conditions are used at all walls ($y = 0$, $y = L_y$ and step's walls). The uniform velocity profile, imposed at the left boundary $x = -L_i$ with $U_o$ being the mean inlet velocity. At outlet, the fully developed condition is used and implemented with a zero gradient for distribution function at the outlet.

## 4. Results and Discussion

Figure 2 shows time average streamlines compared with DNS results. After an initial run of $50T_0$ ($T_0$ is the turn-over time $h/U_o$ and $U_o$=0.05 is the mean inlet velocity), the flow velocity is averaged over time for another $150T_0$. This averaged velocity field is shown in Fig. 2.
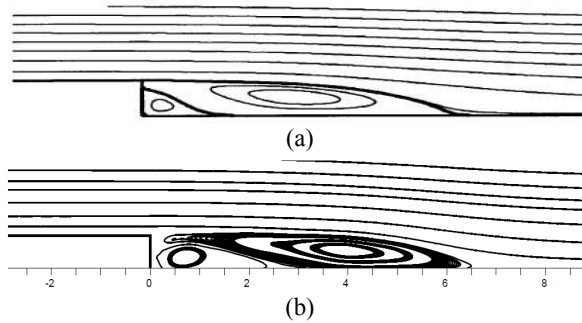


**Fig. 2. Time Average Streamlines in Vertical $z = 0$ Plane,
(a) DNS Results, (b) Current Work.**

Reattachment length is a commonly used parameter to determine the ability of a turbulence model to correctly simulate the flow over backward facing step [11-12]. A key measurement of the computational accuracy of any numerical scheme is the prediction of the reattachment point. This parameter is the distance from the step to the position on the wall, at the bottom of the channel, at which the velocity along the channel becomes positive.

Two options available for the estimation of the reattachment length which are, 1) measuring it with the help of a scale from a simulation result of axial velocity, 2) by defining a line in the numerical simulation, which is very close to the wall and then plotting the variation of axial velocity and by finding the zero streamwise wall shear stress location. The first technique gives a rough estimate of reattachment length and the second technique gives very precise results. Table 1 shows the comparison of the reattachment length obtained by different researches.

**Table 1. Comparisons of Reattachment Length.**

| Research Method | Reattachment length ($Xr/h$) |
|---|---|
| Current work | 6.25 |
| DNS solution by Le et al. [12] | 6.28 |
| Experiment by Jovic and Driver [18] | 6.05 |

(a) *x/h* = 4
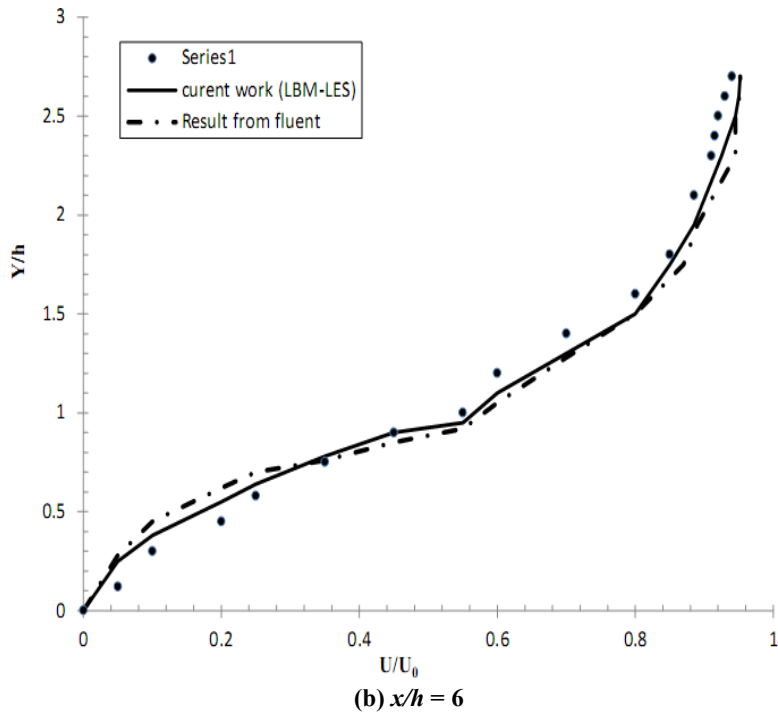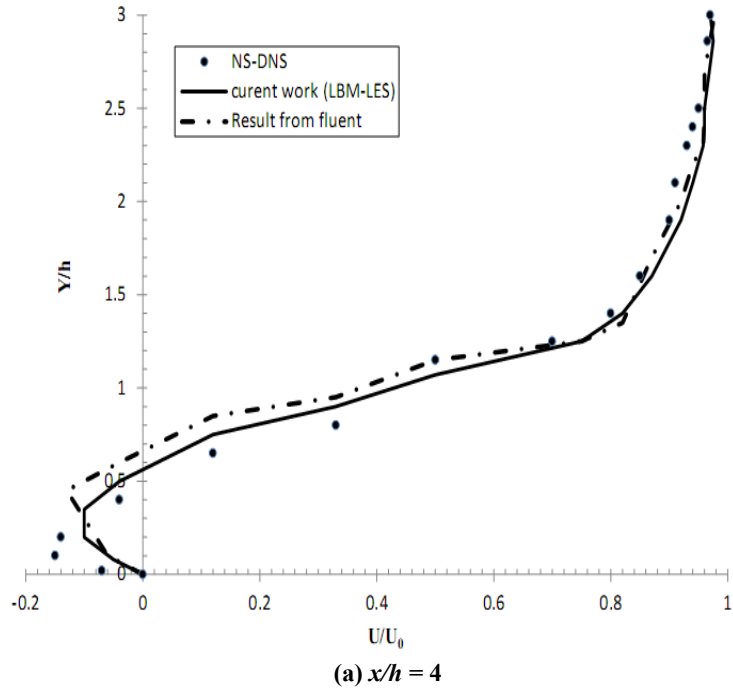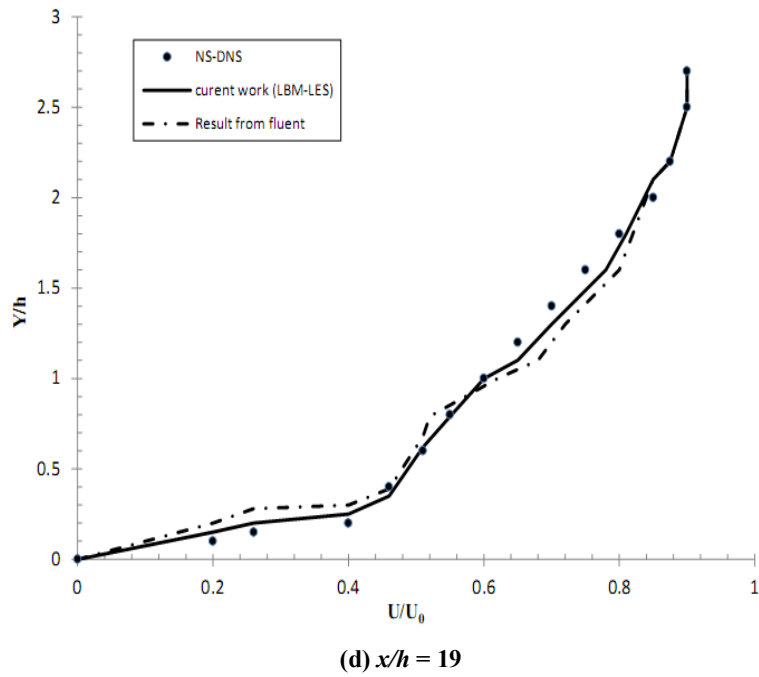


(b) *x/h* = 6
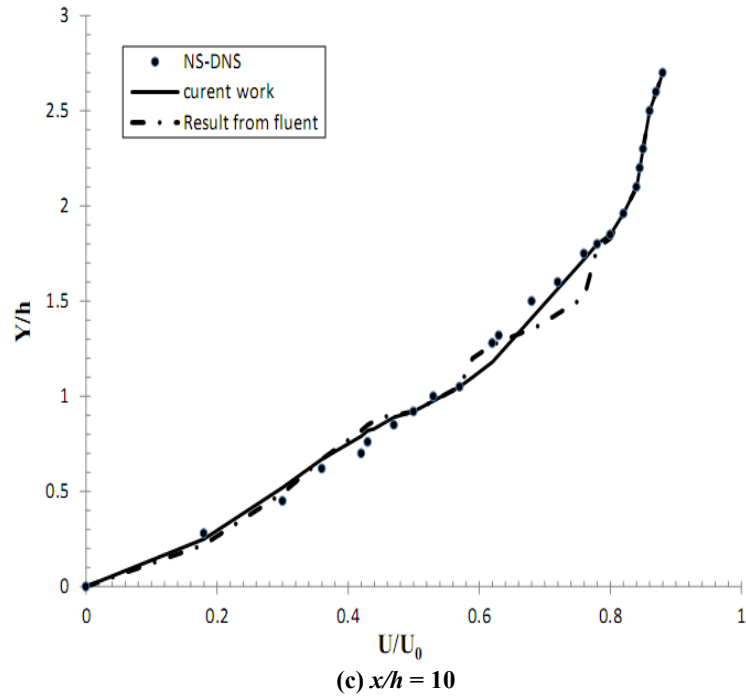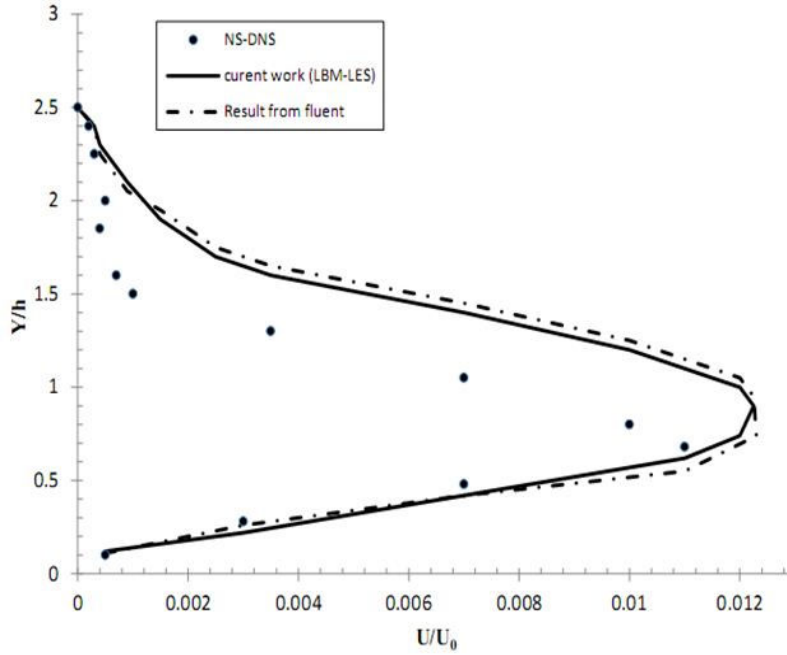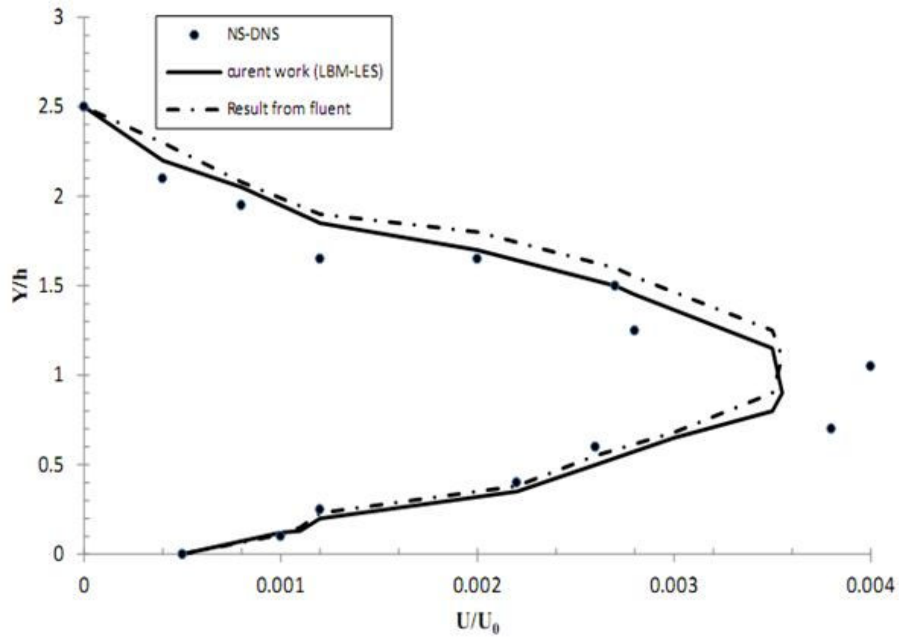
**(c)** *x/h* = 10



**(d)** *x/h* = 19

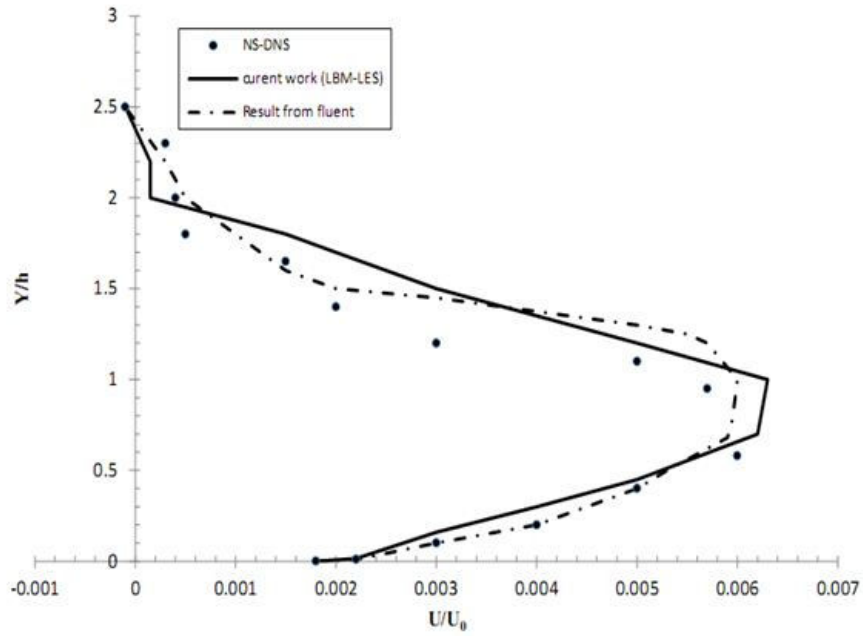**Fig. 3. Time Comparison of Mean Streamwise *U/U*$_o$ Velocity Profiles.**

**(a)** *x/h* = 4



**(b)** *x/h* = 6
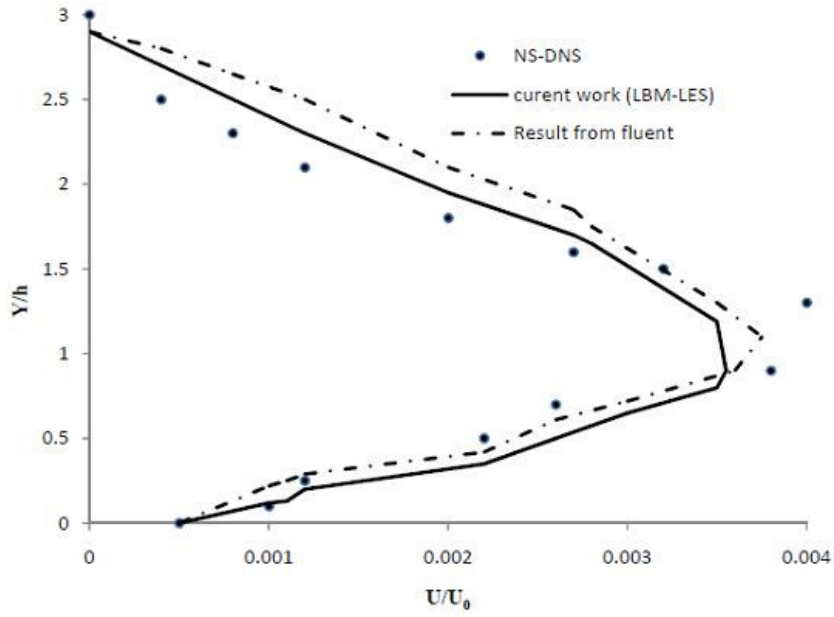
**(c)** *x/h* = 10



**(d)** *x/h* = 19

**Fig. 4.  Comparison of Reynolds Shear stress Profiles at *x/h*.**

The mean reattachment length in current work is 6.25$h$. The *Xr* measurements in Jovic and Driver's experiment [18] vary between 6.0$h$ and 6.2$h$ and *Xr* obtained from DNS solution by Le et al. [12] is about 6.28$h$. Thus, there is a variation of between 2% and 3% among all measured and computed *Xr*. The reattachment length was demonstrated by Kuehn [19] to increase as the expansion ratio increases. Durst and Tropea [20] compiled data from a number of experiments to show that the most dramatic change was in the range $1.10 < ER < 1.30$ where *Xr* varied from 5$h$ to 7$h$. The *Xr* from the current simulation concurs with these findings.

Figure 3 presents the comparison between our simulation with LBM-LES and FLUENT and the DNS for the mean streamwise velocity profiles at x-stations throughout the domain behind the step. The comparison is made at representative locations in the recirculation, reattachment and recovery regions. Excellent agreements between LBE-LES and N.S-DNS results are obtained at all locations. In the recovery region, the velocity has an inflection point at $x/h = 19$ indicating that an equilibrium boundary layer profile is not yet developed.

The Reynolds shear stress component ($u'v'$) is compared with the DNS results at three streamwise locations in Fig. 4. The agreements between computation and the DNS results are excellent for the Reynolds shear stress component. The above quantity was non-dimensionalised with inflow free stream velocity $U_o$.

## 5. Conclusions

Large Eddy Simulation approach was implemented into the lattice Boltzmann method. Three Subgrid Smagorinsky model was carried out for a flow over the backward-facing step at moderate Re=5100. The LBE-LES showed good agreement with the existing DNS data by Le et al. The reattachment length in the longitudinal direction was 6.25$h$ which conforms to DNS result (6.28$h$). The mean longitudinal velocity profile, vertical velocity profile and the Reynolds shear stress compared satisfactory with DNS data set Le et al. All of these profiles conform to DNS results. The work has also been done using Fluent and it showed almost similar results with the other two.

## References

1. Frisch, U.; Hasslacher, B.; and Pomeau, Y. (1986). Lattice-gas automata for Navier-Stokes equation. *Physics Review Letter*, 56(14), 1505-1508.

2. Yu, D.; Mei, R.; Luo, L.S.; and Shyy, W. (2003). Viscous flow computations with the method of lattice Boltzmann equation. *Progress in Aerospace Sciences*, 39(1), 329-367.

3. Munir, F.A.; Azwadi, C.S.N.; and Izual, N.I.N. (2011). Numerical simulation of natural convection in an inclined square cavity. *Journal of Applied Sciences*, 11(2), 373-378.

4. Nor Azwadi, C.S.; Nor Hamizan, M.S.; and Ammar, N.M. (2010). Assessment of lattice Boltzmann simulation scheme in predicting two-phase (solid-fluid) flow. *Jurnal Mekanikal*, 31(1), 11-16.

5.  Yu, H.; Luo, L.S.; and Girimaji, S.S. (2006). LES of turbulent square jet flow using an MRT lattice Boltzmann model. *Computers & Fluids*, 35(8), 957-965.

6.  He, X.; and Luo, L.S. (1997). Lattice Boltzmann model for the incompressible Navier-Stokes equation. *Journal of Statistical Physics*, 88(3), 927-944.

7.  Kolmogorov, A. (1942). *The equations of turbulent motion in an incompressible fluid*, Izv Science.

8.  Chen, S.; Hou, S.; Sterling, J.; and Doolen, G.D. (1996). *A lattice subgrid model for high Reynolds number flows*, Fields Institute Communications.

9.  Teixeira, C. (1998). Incorporating turbulence models into the lattice Boltzmann method. *International Journal of Modern Physics C*, 8(8), 1159-1175.

10. Eggels, J. (1996). Direct and large-eddy simulations of turbulent fluid flow using the lattice Boltzmann scheme. *International Journal of Heat and Fluid Flow*, 17(3), 307-323.

11. Mizukami, M.; Georgiadis, N.J.; and Cannon, M.R. (1993). A comparative study of computational solutions to flow over a backward-facing step. *The Fifth Annual Thermal and Fluids Analysis Workshop*, 523-530.

12. Le, H.; Moin, P.; and Kim, J. (1997). Direct numerical simulation of turbulent flow over a backward-facing step. *Journal of Fluid Mechanics*, 330(1), 349-374.

13. Mussa, M.A.; Abdullah, S.; Nor Azwadi, C.S.; and Zulkifli, R. (2011). Lattice Boltzmann simulation of cavity flows at various Reynolds numbers. *International Review on Modelling and Simulations*, 4(4), 1909-1919.

14. Chapman, S. (1970). *The mathematical theory of nonuniform gases*. Cambridge University Press.

15. Shahrul Azmir, O.; and Nor Azwadi, C.S. (2009). UTOPIA finite different lattice Boltzmann method for simulation natural convection heat transfer from a heated concentric annulus cylinder. *European Journal of Scientific Research*, 38(1), 63-71.

16. Humie`res, D.; Ginzburg, I.; Krafczyk, M.; Lallemand, P.; and Luo, L.S. (2002). Multiple-relaxation-time lattice Boltzmann models in three-dimension. *Philosophical Transaction of the Royal Society of London A*, 360(1792), 437-451.

17. Skordos, P.A. (1993). Initial and boundary conditions for the lattice Boltzmann method. *Physical Review E*, 48(6), 4823-4842.

18. Jovic, S.; and Driver, D.M. (1995). Reynolds number effects on the skin friction in separated flows behind a backward facing step. *Experiments in Fluids*, 18(6), 464-467.

19. Kuehn, D.M. (1980). Some effects of adverse pressure gradient on the incompressible reattaching flow over a rearward-facing step. *Journal of AIAA*, 18(1), 343-344.

20. Durst, F.; and Tropea, C. (1981). Turbulent backward-facing step flows in two-dimensional ducts and channels. *Proceeding of the Third International Symposium on Turbulent Shear Flows*, University of California, 18.1-18.5.