PERIODICA POLYTECHNICA SER. MECH. ENG. VOL. 42, NO. 1, PP. 47–56 (1998)

LARGE-EDDY SIMULATION OF TURBULENT PLANE COUETTE FLOW

H. I. ANDERSSON*, R. KRISTOFFERSEN* and Tamás KALMÁR-NAGY**

*Applied Mechanics Group
The Norwegian University of Science and Technology
N–7034 Trondheim, Norway
**Department of Applied Mechanics
Technical University of Budapest
H–1521 Budapest, Hungary
E-mail: nagykhan.tam.cornell.edu

Received: April 29, 1997

Abstract

The purpose of this study was to explore the central core region of a plane turbulent Couette flow by means of large-eddy simulations. First it was demonstrated how accurately a low Reynolds number flow could be simulated. After having verified the reliability of the LES approach, simulations were performed at a substantially higher *Re*. It was observed that the mean velocity exhibited a practically linear variation in the core region. The extent of the core increased with *Re*, whereas the slope of the mean velocity profile was significantly reduced.

Keywords: large-eddy simulation, Couette-folw, turbulence.

1. Introduction

The shear-driven motion of an incompressible fluid bounded by two parallel walls in relative motion, known as plane Couette flow, is still a problem of practical interest in contexts varying from lubrication theory to thin-film coating. This flow has therefore been studied experimentally for four decades. However, due to difficulties in establishing a reliable and fully developed flow field, the scatter in experimental data sets is disturbing. On the other hand, fully developed flows, in which the streamlines of the mean field are parallel, are particularly amenable to exploration by means of direct large-eddy simulations.

To this end a series of DNS at Reynolds number $Re = \frac{U_w h}{2\nu}$ equal to 1300 was performed by BECH & ANDERSSON [1]. Here, U_w is the velocity difference between the walls, which are separated at a distance $2h$. An extensive comparison between the most convincing computer simulation and new laboratory measurements was provided by BECH et al. [3]. It was found that the near-wall region of the turbulent Couette flow closely resembled those of pressure-driven Poiseuille flow and turbulent boundary layers. The central core region, which typically occupied 50 per cent of the channel cross section, exhibited a nearly uniform mean shear rate. Unfortunately, firm conclusions cannot be drawn from the DNS due to the relatively limited extent of the nearly uniform the central region (core).

In Couette flow at a higher Re, however, the relative extent of the central core, as compared to the near-wall region, is greater, thereby enabling a more thorough exploration of this part of the flow. Since DNS is not feasible at higher Re, the large-eddy simulation approach is adopted to achieve this goal. First, however, a series of LESs have been performed at the lower Re = 1300 in order to demonstrate the reliability of LES.

2. Analytical Background

Fig. 1 shows the configuration for plane Couette flow, where the incompressible fluid is bounded by two infinite plates $2h$ distance apart, and the upper plate moves with velocity U_w, while the lower one is kept fixed. The streamwise length of the computational domain is L_x, while the spanwise length is L_y.

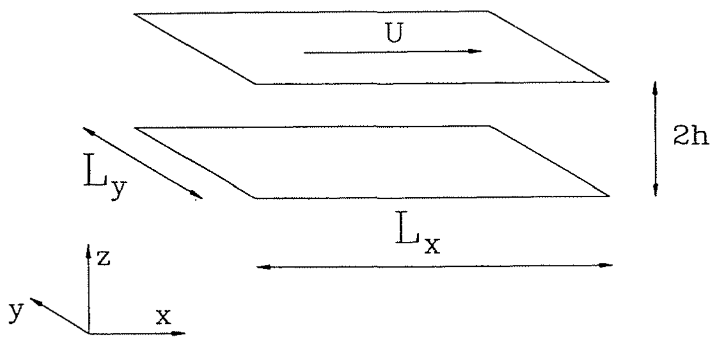

Fig. 1. Plane Couette flow

The dimensionless governing equations are

$$\nabla \cdot \mathbf{u} = 0, \tag{1}$$

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \frac{1}{Re}\Delta \mathbf{u}, \tag{2}$$

where $\mathbf{u} = (u, v, w)$ and Re is the Reynolds number, written in terms of the characteristic scales as

$$Re = \frac{U_w h}{2\nu}.$$

The first boundary condition is the no-slip condition at the walls

$$\mathbf{u}(x, y, -h) = \mathbf{0}, \tag{3}$$
$$\mathbf{u}(x, y, +h) = (U_w, 0, 0). \tag{4}$$

In the homogeneous directions we impose periodic boundary conditions

$$\mathbf{u}(0, y, z) = \mathbf{u}(L_x, y, z), \tag{5}$$
$$\mathbf{u}(x, 0, z) = \mathbf{u}(x, L_x, z). \tag{6}$$

3. Computational Approach

The three-dimensional unsteady Navier-Stokes equations were solved numerically using the Explicit Channel Code for Large-Eddy Simulation developed by GAVRILAKIS et al. [5]. ECCLES is a finite-difference code with second-order central-diffrence approximations in space and a second-order Adams–Basforth scheme in time. The primitive variables are positioned in a staggered grid arrangement.

The computational domain was $4\pi h \times 2\pi h$ in the streamwise (x) and spanwise (z) directions, respectively, while the height was $2h$.

The initial velocity field was constructed of the laminar mean velocity profile $U/U_w = (z+h)/2h$ with random velocity fluctuations superimposed.

From this unphysical velocity field the program was running until the flow reached a statistically steady state.

This statistically steady fully developed turbulent Couette flow was identified by its property that

$$\mu \frac{du}{dz} - \rho \overline{u'w'} = \tau_{wall}, \tag{7}$$

i.e. the sum of the viscous shear stress and the Reynolds shear stress is constant across the entire channel (this can easily be derived from the Reynolds averaged Navier-Stokes equations).

Then the ensemble of statistically independent samples was gathered with further simulations.

Different computational grids and three sub-grid scale (SGS) models were considered, i. e. those due to Smagorinsky, Schumann and Moin & Kim.

The gridpoints were uniformly distributed in the x- and y-directions, while uniform or tanh distribution was used in the inhomogeneous z-direction (the latter to improve the resolution in the near-wall regions).

4. The Effect of Grid Resolution and Subgrid-Scale Closures

Fig. 2 shows mean velocity profiles from direct simulations (i.e. without SGS model) at Reynolds number 1300 and 7050 (left and right figures, respectively).

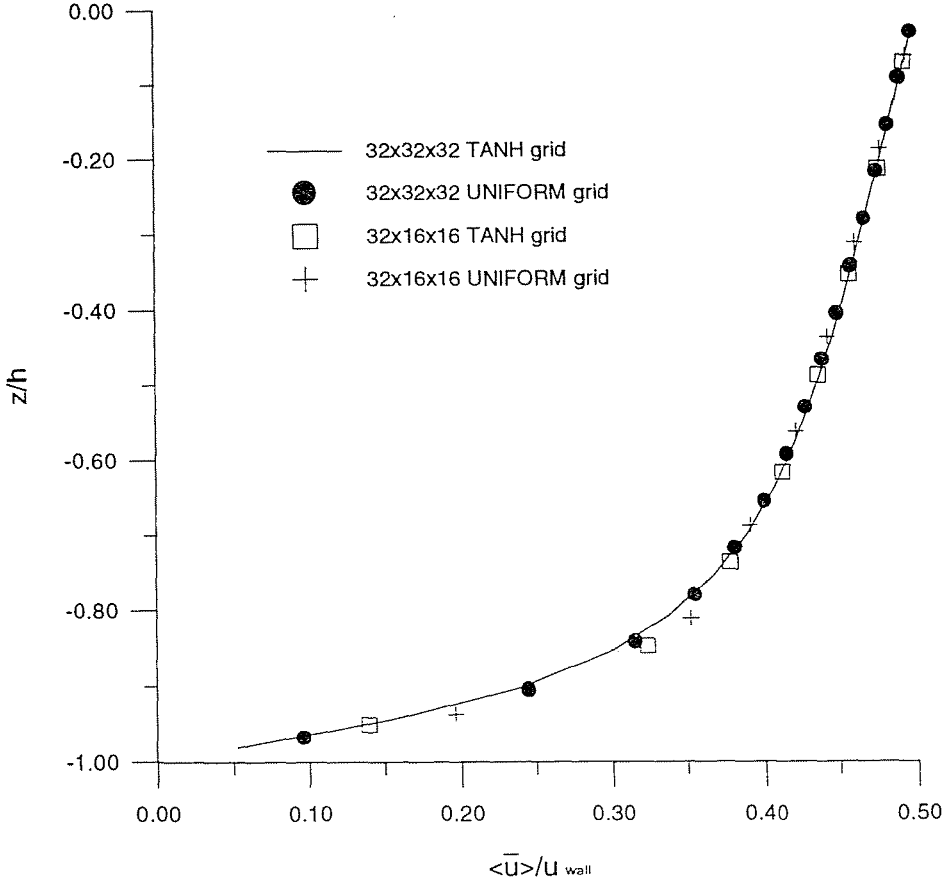

Fig. 2a The effect of grid resolution on the mean velocity profile

In the central region the mean velocity gradient is fairly the same for all the used grids (but different for different Reynolds numbers), however, at the significantly higher Reynolds number $Re = 7050$, the simulated flow field was substantially affected by the grid resolution, especially in the near-wall region.

At the lower Reynolds number the results from simulations without SGS model were quite similar to those obtained with models.

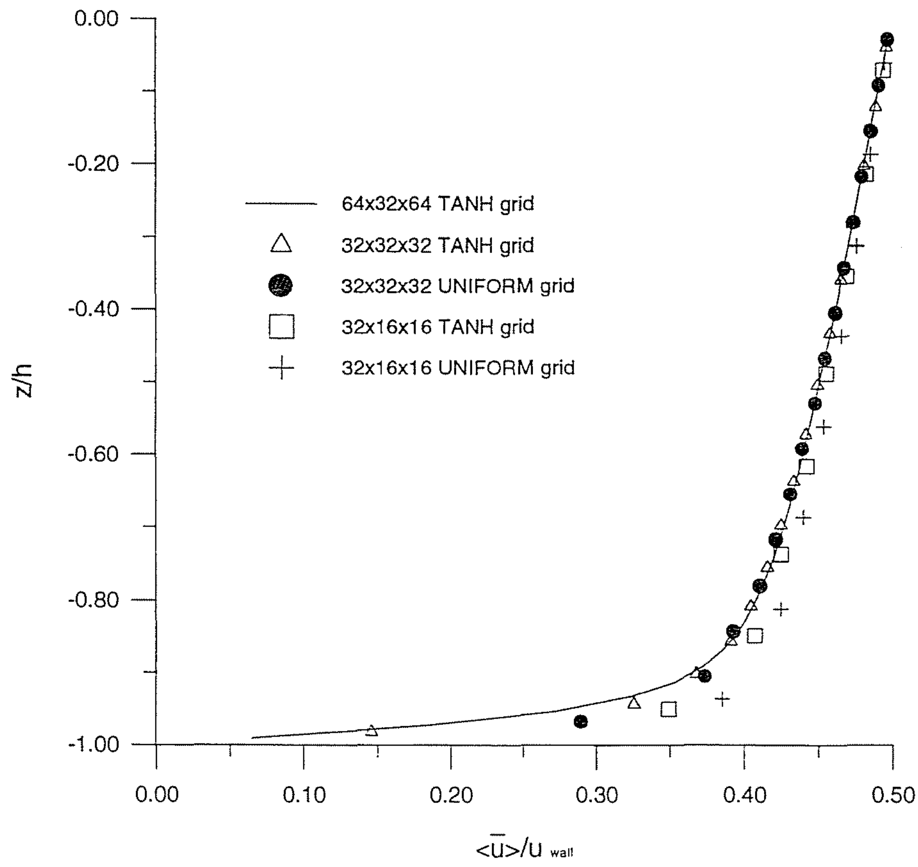

Fig. 2b The effect of grid resolution on the mean velocity profile

It was shown by MIYAKE et al. [7] that the Smagorinsky model is not adequate for modelling Couette flow, because of the high anisotropy of the flow near the walls. This was also confirmed during the simulations. The Schumann's model underpredicted the velocities at the higher Re, while Moin & Kim's model turned out be the most appropriate.

From here on we only consider the results from the simulations with $32 \times 32 \times 32$ and $64 \times 32 \times 64$ streched grids (for $Re = 1300$ and $Re = 7050$, respectively). In *Fig. 3* we compare the mean profile and intensities (root mean square value of the turbulent velocity in each direction) for $Re = 1300$ from our simulation with the one measured by AYDIN & LEUTHEUSSER [1] and the most reliable DNS available [3].

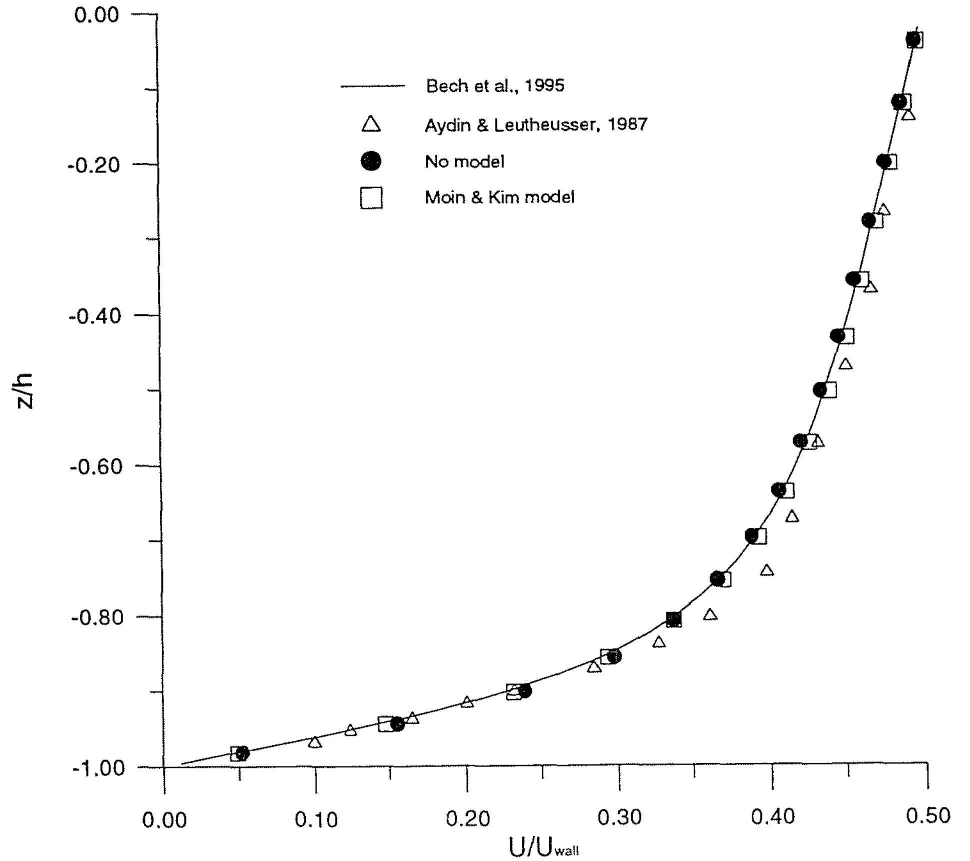

Fig. 3a Comparison of simulations and experiment (mean profile and intensities) $Re = 1300$

The agreement is good, however, the experimental velocity profile has a generally steeper profile. It is surprising that the profile of our simulation matches the curve from the fully resolved simulation of Bech (with $256 \times 70 \times 256$ gridpoints) very well. This means that the resolved scales are responsible for most of the turbulent transport. So, the underlying idea of LES is thus confirmed by these results.

The intensity of the velocity fluctuation is dominant in the streamwise direction. The model slightly overpredicts the streamwise fluctuation. The higher Reynolds number is sufficiently high for the unresolved scales to contain important contribution. The effect of these scales is accounted for by the SGS model.

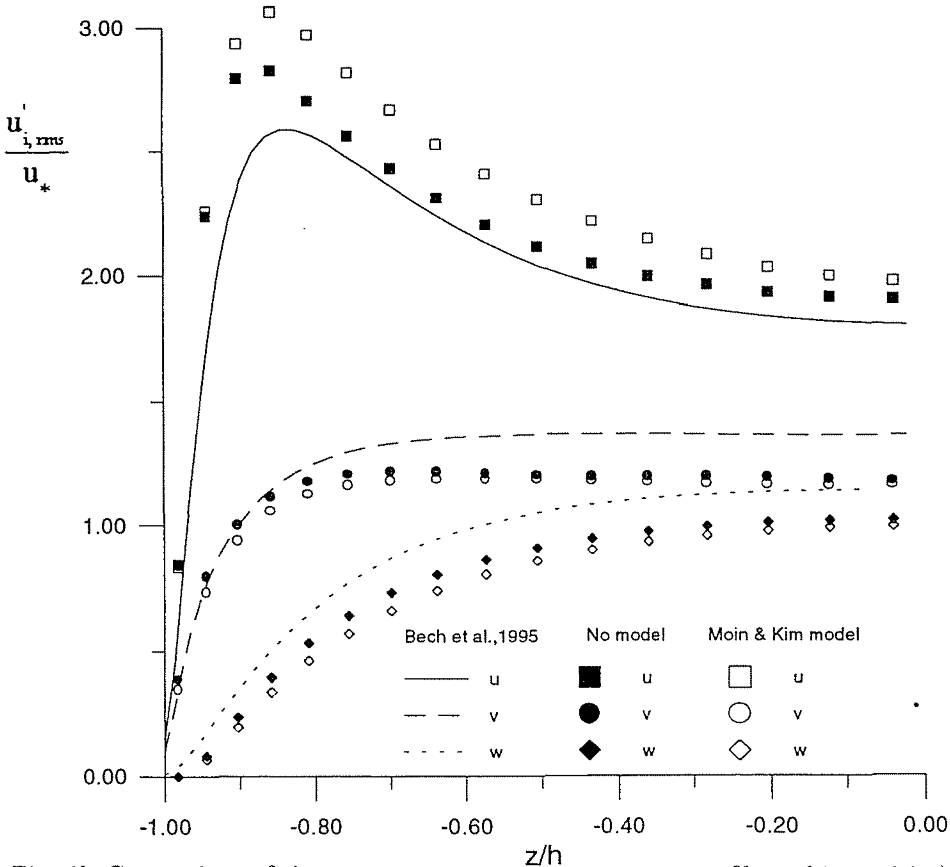

Fig. 3b Comparison of simulations and experiment (mean profile and intensities) $Re = 1300$

The mean profile and turbulence intensities for this higher Re case from 'direct' and model simulations are shown in *Fig. 4*. In their experiment ROBERTSON & JOHNSON [9] found a peak level of u of about 2.7, i.e. below the peak level obtained in the LES. However, scaling of experimental data with the friction velocity may cause errors, since this quantity is difficult to accurately determine experimentally.

5. Discussion

REICHARDT [8] was the first to observe that the plane Couette flow exhibited a core region with nearly linear variation of the mean velocity U and that the slope $S = \frac{2h}{U_{wall}} \frac{dU}{dz}$ of the mean velocity profile $U(z)$ decreased with increasing Reynolds number. In the limit of infinitely high Re BUSSE [4] found a lower bound S =0.25 for this slope. Rather surprisingly, the DNS of BECH et al. [3] showed that $S = 0.22$ at $Re = 1300$, i.e. slightly below the theoretically established lower limit. It is therefore noteworthy that the LES for $Re = 1300$ gives an S-value of 0.20, i.e. in good agreement with the directly simulated Couette flow. When the Reynolds number is increased from 1300 to 7050, the extent of the core region, defined as the range in which the mean velocity U does not deviate more than 0.01 U_w from the tangent line at $z = 0$, increased from 51% to 64% of the channel width $2h$. Even more interesting is the observation that the dimensionless slope reduced to $S = 0.17$, i.e. some 30 per cent below the theoretically determined lower bound.

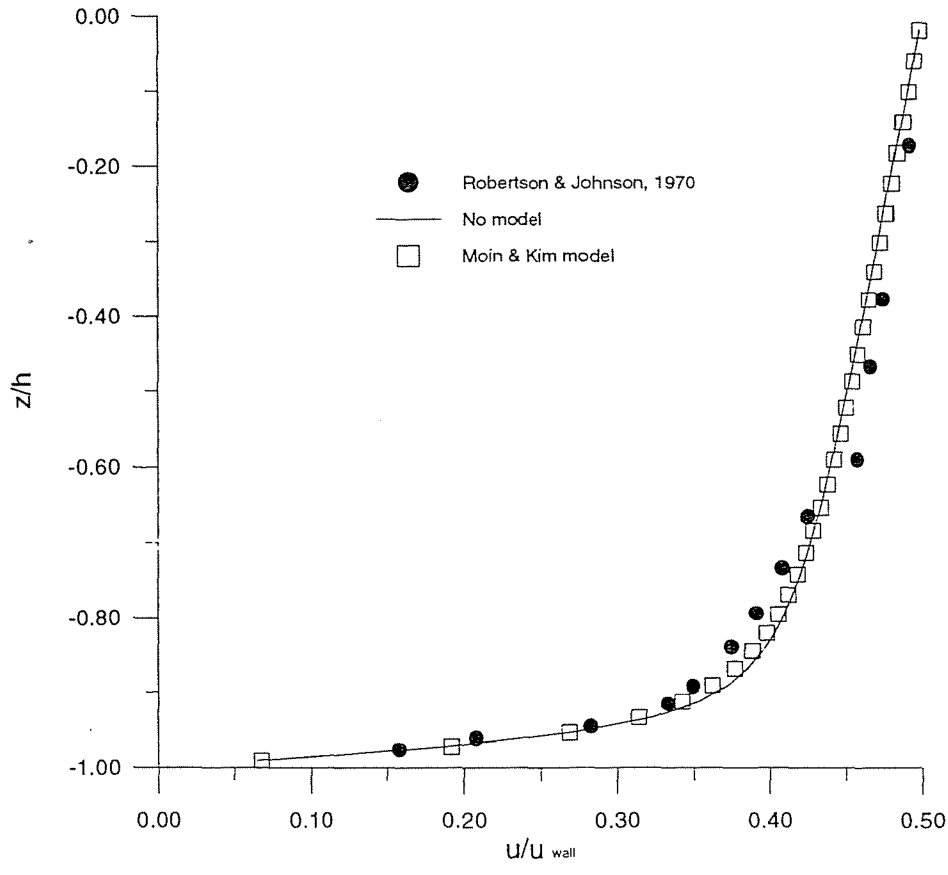

Fig. 4a Comparison of simulations and experiment (mean profile and intensities) $Re = 7050$

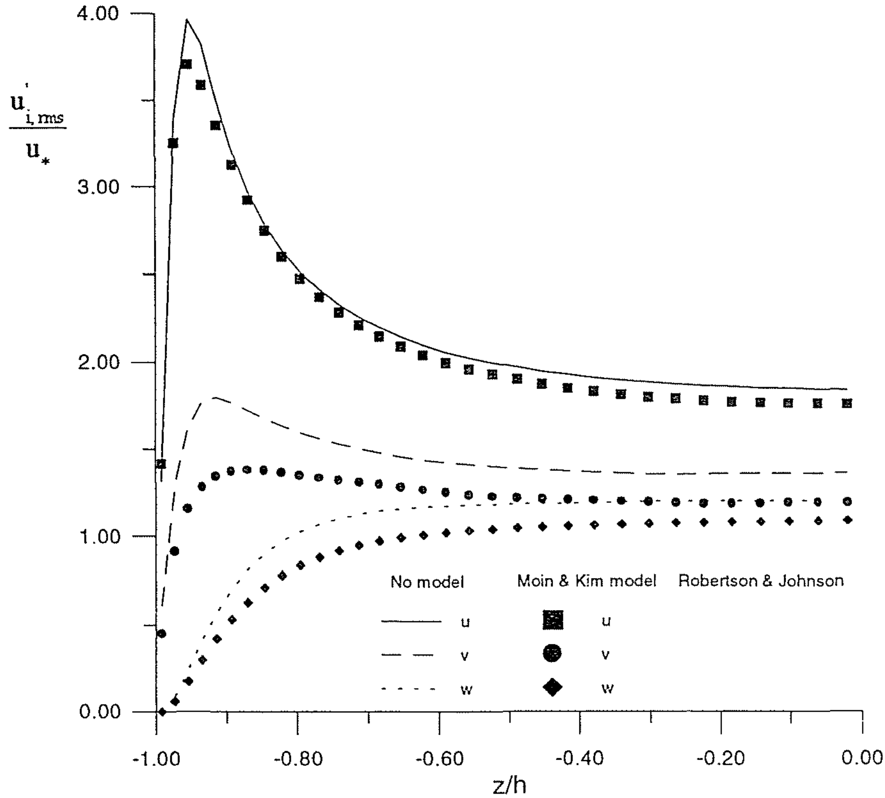

Fig. 4b Comparison of simulations and experiment (mean profile and intensities) $Re = 7050$

Moreover, a significant part of the mean velocity profile in *Fig. 4* exhibits a nearly increasing velocity U in contrast with the inverse cosine variation deduced by VON KÁRMÁN [6].

Equally interesting is the observation that the slope of the mean velocity profile is about 20% below the theoretically established lower bound for the limit of infinite Re [4]. It is moreover noteworthy that the region with linearly increasing U is associated with a constant turbulent shear stress $-\rho\overline{u'w'}$ (sum of resolved and SGS-modelled part).

The local turbulence intensity at the centreline is much higher than the turbulence intensity at the centreline in plane Poiseuille flow. This is probably because of the fact that for plane Couette flow the turbulence production is not zero at the centreline.

6. Conclusions

This study has shown that the large-eddy simulation provides a useful approach to plane Couette flow at intermediate Reynolds numbers which cannot be directly simulated due to the capacity of present days supercomputers. The reliability of the present LES seems to compare favourably with most laboratory investigations. The simulations confirmed the conjecture that the mean velocity gradient in the core region decreases with the increasing Reynolds number.

7. Acknowledgement

T. Kalmár-Nagy would like to acknowledge the grant from the Norwegian University of Science and Technology for his stay in Trondheim and the computer time on a CRAY-YMP generously provided by SINTEF.

References

[1] AYDIN, E. M. – LEUTHEUSSER, H. J. (1987): Experimental Investigation of Turbulent Plane Couette Flow, *ASME Forum on Turbulent Flows* FED Vol. 51, Cincinnatti.

[2] BECH, K. H. – ANDERSSON, H. I. (1994): In Direct and Large-eddy Simulation I, p. 13.

[3] BECH, K. H. – TILLMARK, N – ALFREDSSON, P. H. – ANDERSSON, H. I.(1995): An Investigation of Turbulent Plane Couette Flow at Low Reynolds Numbers, *J. Fluid Mech.*, Mar. 10, 1995, pp. 286, 291–325.

[4] BUSSE, F. H. (1970): *J. Fluid Mech.*, Vol.41, p. 219.

[5] GAVRILAKIS, S. –TSAI, H. M. – VOKE, P. R. – LESLIE, D. C. (1986): Large-eddy Simulation of a Low Reynolds Number Channel Flow by Spectral and Finite Difference Methods, *Notes on Numerical Fluid Mechanics*, Vol. 15, pp. 105–118.

[6] VON KÁRMÁN, T. (1937): *J. Aeron. Sci.* 4, p. 131.

[7] MIYAKE, Y. – KAJISHIMA, T. – OBANA, S.(1987): Direct Numerical Simulation of Plane Couette Flow at a Transitional Reynolds Number. *JSME International Journal*, Vol. 30(259) pp. 57–65.

[8] REICHARDT, H.(1956): Über die Geschwindigkeitsverteilung in einer geradigen turbulenten Couetteströmung, *Zeit. Angew. Math. Mech.* Vol.36, pp. 26–29.

[9] ROBERTSON, J. M. – JOHNSON, H. F.(1970): Turbulence Structure in Plane Couette Flow. *ASCE J. Eng. Mech. Div.* Vol. 96, pp. 1171–1182.