AIR MOVEMENT & VENTILATION CONTROL WITHIN BUILDINGS

12th AIVC Conference, Ottawa, Canada
24-27 September, 1991

POSTER 45

Numerical Investigation of Transient Flow Over a Backward Facing Step Using a Low Reynolds Number k-e Model.

M. Skovgaard, P.V. Nielsen

University of Aalborg
Denmark

NUMERICAL INVESTIGATION OF TRANSIENT FLOW OVER A BACWARD FACING STEP USING A LOW REYNOLDS NUMBER k-ε MODEL.

By

M.Skovgaard and P.V.Nielsen,
The University of Aalborg, DK

SUMMARY.

Recent full scale experiments has detected the presence of low Reynolds number effects in the flow in a ventilated room. This means that one are unable to predict the flow patterns in some geometries for air change rates - or Reynolds numbers - which are relevant for ventilation engineering by a standard model of turbulence.

In this paper it is investigated if it is possible to simulate and capture some of the low Reynolds number effects numerically using time averaged momentum equations and low Reynolds number k-ε model. The test case is the laminar to turbulent transistional flow over a backward facing step with expansion ratio (h/H eq. 1/6).

The results are evaluated and held up against experimental LDA data and simulation with a Reynolds stress model (RSM).

ACKNOWLEDGEMENTS.

LIST OF SYMBOLS

$C_{\epsilon 1}$	Constant in the turbulence model
$C_{\epsilon 2}$	Constant in the turbulence model
$C_{\epsilon 3}$	Constant in the turbulence model
C_{μ}	Constant in the turbulence model
E	Wall roughness function in the logarithmic law
f_{μ}	Function which mimics the direct effect of the molecular viscosity on the shear stress in the turbulence model
f_1	Function to increase the dissipation near the wall in the turbulence model
f_2	Function to incorporate low Reynolds number effects in the destruction term of the ε equation
h	Inlet height
H	Channel height
LRN	Low Reynolds number
k	Turbulent kinetic energy
P	Pressure, generation term
R	Turbulent Reynolds number
Re	Reynolds number ($\rho l U/\mu$)

RSM	Reynolds stress model
S	Source term
u,v,w	Velocity fluctuations
U,V,W	Mean velocities
U^+	Dimensionless velocity parallel to the surface ($U_p/U\tau$)
x,y,z	Directions
y	Normal distance from wall
y+	Dimensionless wall distance ($U_\tau y_p \rho/\mu$)

Subscripts

i,j,k	Indicators of direction
o	Inlet
RE	Recirculation
S	Shear
t	Turbulent

Greek

δ	Kronecker delta, area
ϵ	Energy dissipation
$\underline{\epsilon}$	Energy dissipation in the Launder-Sharma k,ϵ model
$\varkappa$	von Karman constant
μ	Viscosity (dynamic)
ρ	Density
τ	Shear stress
σ	Constant in the turbulence model (the turbulent Prandtl number)
ϕ	Generalized variable

INTRODUCTION

The flow patterns in mechanically ventilated rooms gives arise to many complications when one wants to predict them theoretically and/or numerically. These complications ranges from the fact that the flow is turbulent and the confined space is relatively large to the complexcities of the geometrical design of the components involved. Factors as transitional flow through inlet devices, transistional effects in the resulting jets and low Reynolds number effects in the room where the velocities are low. are also very important.

The succes of numerical predictions in this area depend very much of the situation. If one or more of the above mentioned factors are involved - which is often the case - then the numerical procedure and the mathematical models must be able to capture these phenomenas.

Some studies of the complicated factors has recently been carried out - both numerically (e.g. *Skovgaard et. al. 1991a, Murakami, 1983* and *Chen 1990*) and experimentally (e.g. *Nielsen et al. 1988, Skovgaard el. al. 1990 , Heiselberg et. al. 1987* and *Restivo 1979*).

Present paper reports work done on the low Reynolds number flows near the wall and in the transitional jet regime. Both areas are important when one wish to predict flow patterns in a ventilated room because the velocity level in a room is strongly influenced by the inlet momentum flow and - at lower velocities - boundary layer flow.

In order to be able to separate the two subjects from other complexities which might

occur in a real situation a simpler geometry is adopted. The geometry of a two - dimensional single sided sudden expansion of ratio 1/6 is chosen. The simulation covers a range of Reynolds number which is typical for room air flows. This is the same range which covers the transistional regime with evidence of periodicy (flow experiments *Restivo 1979*) to the high velocity regime where the turbulent fluctuations is spread over a wide frequency range and the Reynolds number dependence is little (fully turbulent region). Comparisons with experiments are available from work done by *Restivo 1979*.

THE MODEL AND THE NUMERICAL APPROACH

The model consist of the continuity - and the momentum equations for the time averaged flow and a eddy viscosity concept for the turbulent Reynolds stresses.

The govering equations for steady flow are

$$\frac{\partial}{\partial x_i}(\rho U_i) = 0 \tag{1}$$

$$\frac{\partial}{\partial x_i}(\rho U_i U_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_i}\left(\mu\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)\right) - \rho\overline{u_i u_j} \tag{2}$$

To describe the Reynolds stresses several models can be adopted. It is here chosen to apply a low - Reynolds number (LRN) form of the k-ϵ model which is still the most used in engineering types of flow. *Patel et. al. 1985* reviewed several forms of LRN k-ϵ models and although differences were recorded several models gave similar solutions in the prediction of the flate plate boundary layer. The Launder - Sharma version of the Jones - Launder model is applied here (*Launder and Sharma 1978*).

The model takes the following form for 2D isotropic homogenious flow

$$\mu_t = \rho C_\mu f_\mu \frac{k^2}{\varepsilon} \; ; \; \underline{\varepsilon} = \varepsilon - 2\frac{\mu}{\rho}\left(\frac{\partial}{\partial x_i}\left(\frac{\partial \sqrt{k}}{\partial x_j}\right)\right)^2 \tag{3}$$

$$\frac{\partial}{\partial x_i}(\rho U_i k) = \frac{\partial}{\partial x_i}\left(\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right) + P - \rho\underline{\varepsilon} \tag{4}$$

$$\frac{\partial}{\partial x_i}(\rho U_i \underline{\varepsilon}) = \frac{\partial}{\partial x_i}\left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right) + C_{\varepsilon 1} f_1 \frac{\varepsilon}{k} P - C_{\varepsilon 2} f_2 \rho \frac{\varepsilon^2}{k} + C_{\varepsilon 3} \frac{\mu \mu_t}{\rho}\left(\frac{\partial}{\partial x_i}\left(\frac{\partial U_i}{\partial x_j}\right)\right)^2 \quad (5)$$

Where P is the generation rate due to shear effects

$$P = -\rho \overline{u_i u_j} \frac{\partial U_i}{\partial x_j}$$

Reynolds stresses are computed from

$$-\rho \overline{u_i u_j} = \frac{2}{3}\delta_{ij}\rho k - \mu_t\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)$$

The model constants and functions can be seen in tabel 1.

It should be noticed that ϵ in (5) is replaced by $\underline{\varepsilon}$ which is the total dissipation rate. This is a convenient form which allows the $\underline{\varepsilon}$ wall boundary conditions to be zero. The trade - off of this substitution is the complex source term $C_{\epsilon 3}$.....

Regarding the calculations of the source terms $C_{\epsilon 1}$..... and $C_{\epsilon 2}$..... the authors found that it should be calculated as written in 5 and if any under-relaxation is needed the k-value from previous iteration can conveniently be used.

The k-ϵ model.	Fully turbulent version.	LRN version.
C_μ	0.09	0.09
$C_{\epsilon 1}$	1.44	1.44
$C_{\epsilon 2}$	1.92	1.92
$C_{\epsilon 3}$	-	2
σ_k	1.0	1.0
σ_ϵ	1.3	1.3
f_μ	1.0	$\exp(-3.4/(1+R_t/50)^2)$
f_1	1.0	1.0
f_2	1.0	$1-0.3\exp(-R_t^2)$

Tabel 1: The turbulent model constants and functions. $R_t = \rho k^2/\mu \underline{\varepsilon}$

NUMERICAL PROCEDURE

The two-dimensional elliptic flow solver TEAM *(Huang 1986)* which has been extentively used and validated was used to obtain the flow solutions. The code is using the finite volume technique to solve the equations mentioned in the previous paragraph imploying the staggered grid layout to overcome the checkerboard phenomenon. The solution scheme in SIMPLE and the differencing scheme was QUICK for convective terms in the momentum eq.s and PLDS for other terms and variables. A non uniform grid was used in order to achieve a finer grid in the near wall region and in the shear layers. In fact the solution is rather sensitive to the grid layout especially the region $5 < y^+ < 30$. The equations was solved line by line in an ADI - iterative manner. As expected the convergence is rather slow because of the slow diffusive proces in the boundary layer and because of the necessary fine grid.

TEST IN STRAIGHT CHANNEL FLOW

To validate the performance of the model it is chosen to apply it to fully developed channel flow. Primarily because a wide range of data is available for comparison and second because the fully developed u, k and $\underline{\epsilon}$ profile will serve as inlet conditions in the later application.

The case was run one-dimensional with fixed dP/dx, in the u - momentum eq. In this way it is only nessesary to solve for u, k and $\underline{\epsilon}$. 35 gridnodes was used in the crossstream direction. Boundary condition for u, k and $\underline{\epsilon}$ was set to zero on the walls.

Fig. 1a depicts the mean velocity up through the boundary layer for $Re_{bulk} = 50,000$. As seen are the U^+ values in the sublayer in good agreement with the RSM data *(Launder et. al. 1990)* and the experimental data of *Laufer 1949*. In the outer - fully turbulent region are the U^+ values too high compared to the log. law and the RSM data which *Patel et. al. 1985* concluded to arise from the source term $C_{\epsilon 3}$..... which

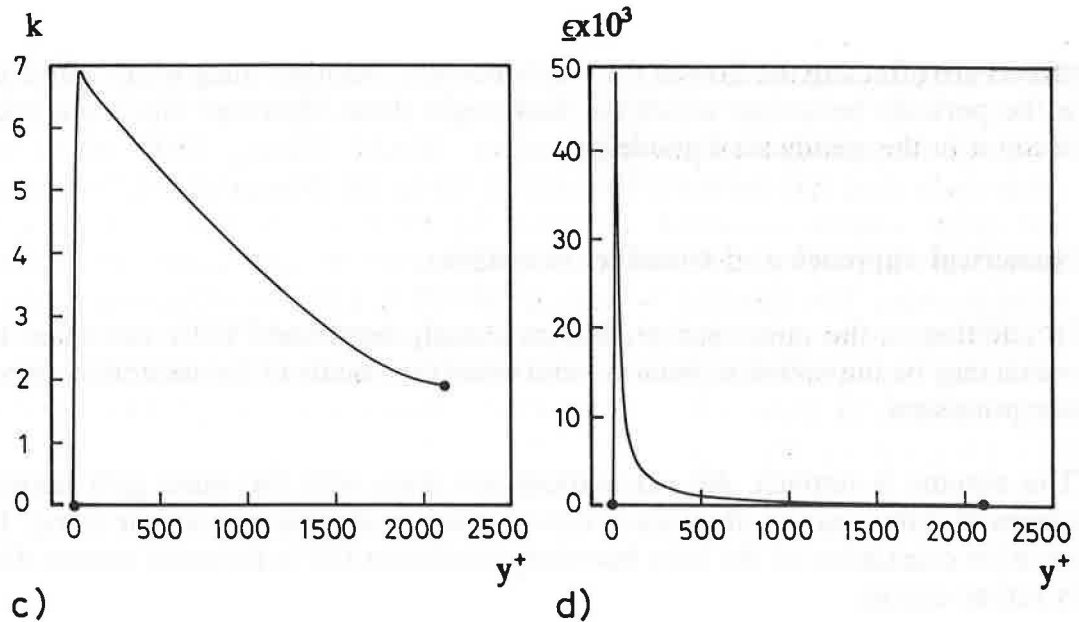a)   b)

206
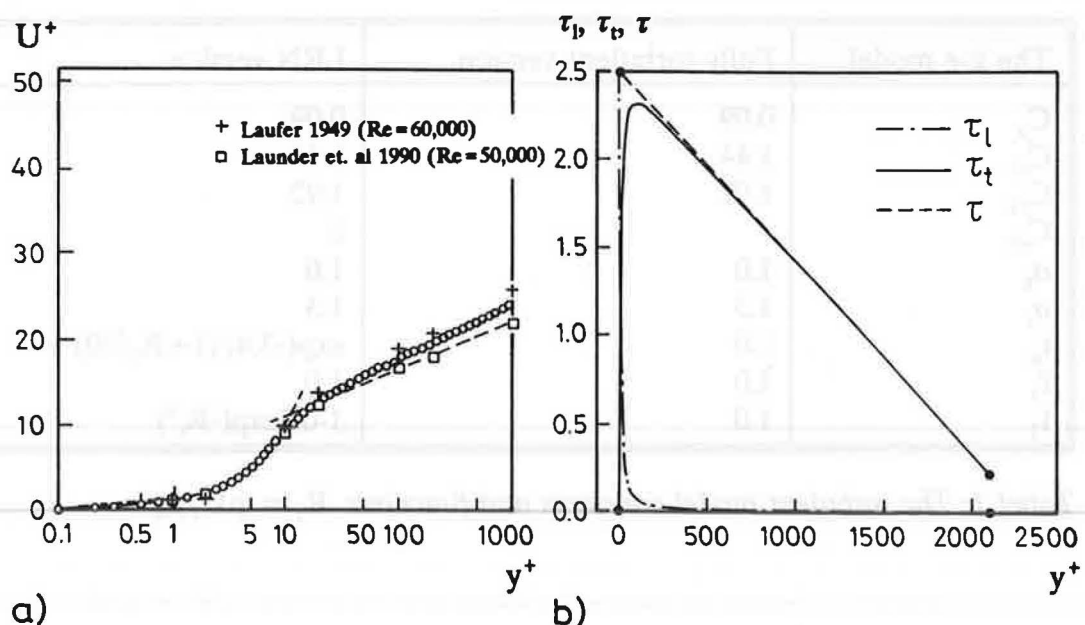
Figure 1. Simulated data from the straight channel flow (line). Experimental data by Laufer 1949 (plus). RSM data by Launder and Tselepedakis 1990 (square). a) U^+ values as a function of y^+, b) shear stress profiles, c) k profiles and d) $\underline{\varepsilon}$ profiles.

increases the dissipation level in the shear layer giving a too low k level.

TRANSIENT CALCULATION IN THE BACKWARD FACING STEP GEOMETRY

In the following paragraph the LRN model is applied in an numerical experiment to see whether it is possible to predict the transient flow over a backward facing step geometry (fig. 2) with a expansion ratio of 1/6 (h/H =1/6) in the Re_{inlet} range of 0-5050.

The interesting thing about this numerical eksperiment is to see if it possible to get a solution in the region 500 < Re < 5000 - which is very important for ventilation engineering. In this region the peak-velocity in the jet and the velocity decay are different from the fully turbulent behaviour and the turbulent viscosity in the recirculation zone is in the same order of magnitude as laminar viscosity. All those
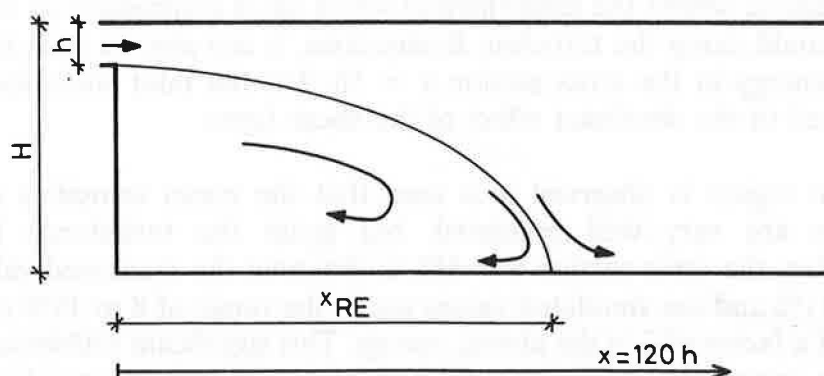

Figure 2. Sketch of backward facing step geometry. h/H = 1/6.

effects are affecting the flow in the whole domain. Another thing which affect the flow is the periodic behaviour which the flow might show. However this is not taken into account in the steady state model.

Numerical approach and boundary conditions

In addition to the numerical procedure already mentioned there are some features which may be important to bear in mind when the results of the numerical experiment are presented.

The scheme is implicit. All calculations are done with the same grid layout which means that the amount of nodes in the sub - and buffer layer is not the same. The grid used for calculation of the inlet boundary conditions (80 in the cross stream direction) is rather coarse.

The boundary conditions are as follows

inlet: u, k, ϵ - calculated profiles by the method mentioned in the previous chapter.
$$v = 0$$

wall: $u = v = k = \epsilon = 0$

outlet: $v = 0; du/dx = dk/dx = d\epsilon/dx = 0$.
The outlet is placed 120h downstream were the flow is expected to be uniform.

RESULTS

Fig 3 shows the results from a simulation with $Re_{bulk} = 5{,}050$. As seen the inlet conditions for U - velocity is well predicted so the inlet momentum is exactly the same as recorded in the experiments. The $k^{1/2}/U_o$ values are on the other hand much higher than the experimental values. The predicted values are in the interval from 7 to 13% where the measured are from 2 to 6%. This discrepancy - which is significant may be caused by differences in inlet conditions. In present simulation the inlet condition is strictly two dimensional where the experimental setup has a contraction in the third dimension which could damp the turbulent fluctuations. It can also be seen from the values of kinetic energy in the cross section x = 5h that the inlet condition is not important compared to the dominant effect of the shear layer.

If the downstream region is observed it is seen that the mean velocities and the recirculation zone are very well predicted, but again the turbulence level is overpredicted. If f.ex. the cross section x = 30h is observed the measured values are in the range 6 to 11% and the simulated values are in the range of 8 to 15% resulting in a discrepancy of a factor of 2 in the kinetic energy. This significant difference in the turbulence level is unexpected taking the good agreement in the mean velocity into account. Also previuos calculations (*Skovgaard et. al. 1991a*) with the same Re -

number in a very similar but confined enclosure has shown that the *Launder - Sharma* model is able to predict the k - level, but one should bear in mind that the relation u eq. $k^{1/2}$ is very dependent on the turbulent flow type. *Restivo 1991* reported also that the flow was very unstable and had a periodic tendency.

The maximum value of u in a fully developed wall jet is close to $0.22U_x$, where U_x is the peak velocity in the profile at a given distance (*Nelson 1969*). If this assumption is used on the flow in fig. 3 it shows that k is underpredicted at $x = 10h$ and 15h while it is over predicted further downstream. All the measurements has a lower level of u compared to the values expected in a fully developed self-similar wall jet.
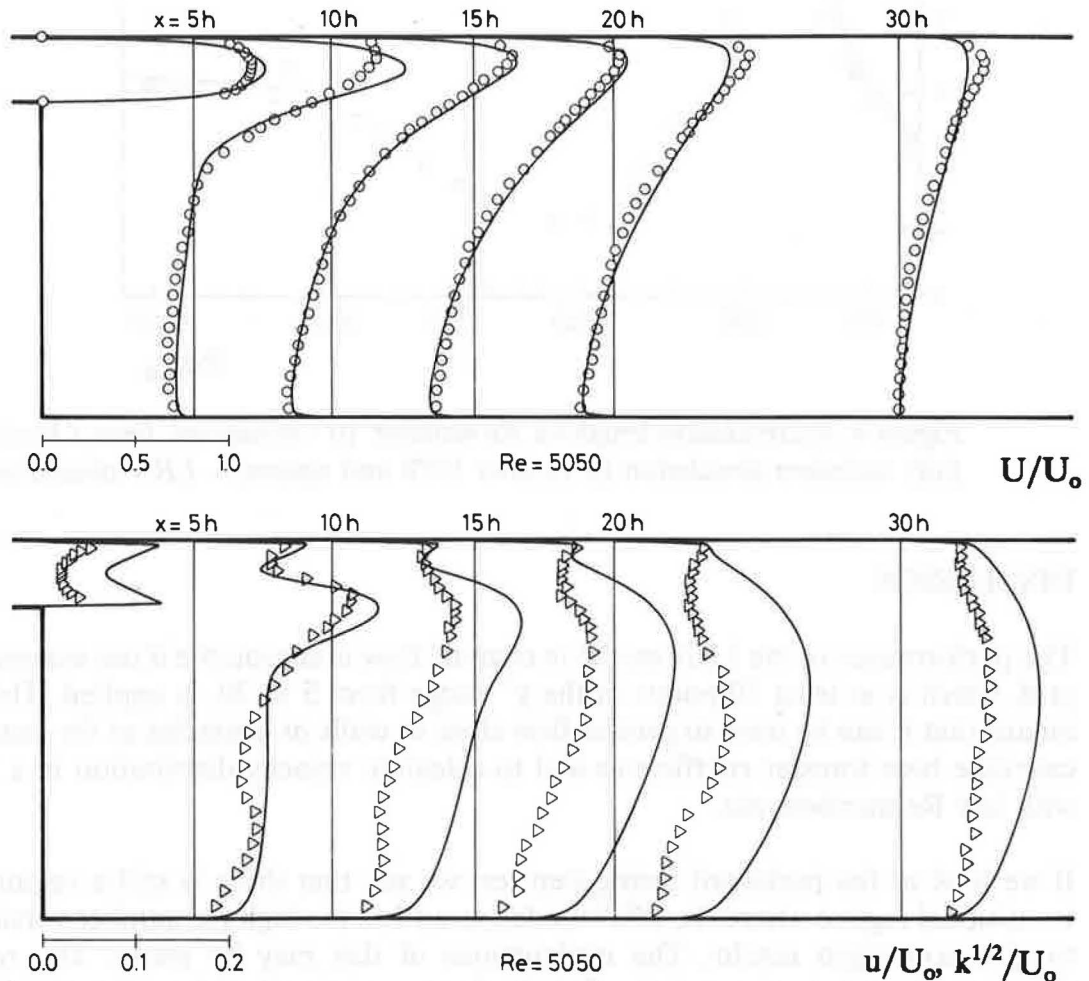

Figure 3 comparisons of present data (line) with experimental values from Restivo (marks). Re = 5,050. a) velocity profiles. b) turbulence intensity.

Figure 4 shows the recirculation length as a function of the Reynolds number. The figure indicates that the mean flow pattern in the backward facing step varies very substantially for different Re numbers. The behaviour has also been reported of several other authors in geometries with other expansion ratios.

The figur shows measured data by *Restivo 1979* compared with simulated results. It is seen that all the numerical models fails to give the same tendency as measured. If we

focus on the LRN results specificly is it seen that there is a region (below Re = 1000) where it is impossible to obtain a converged solution, because the function f_μ is close to zero and the model consequently is equal to the laminar set of equations.
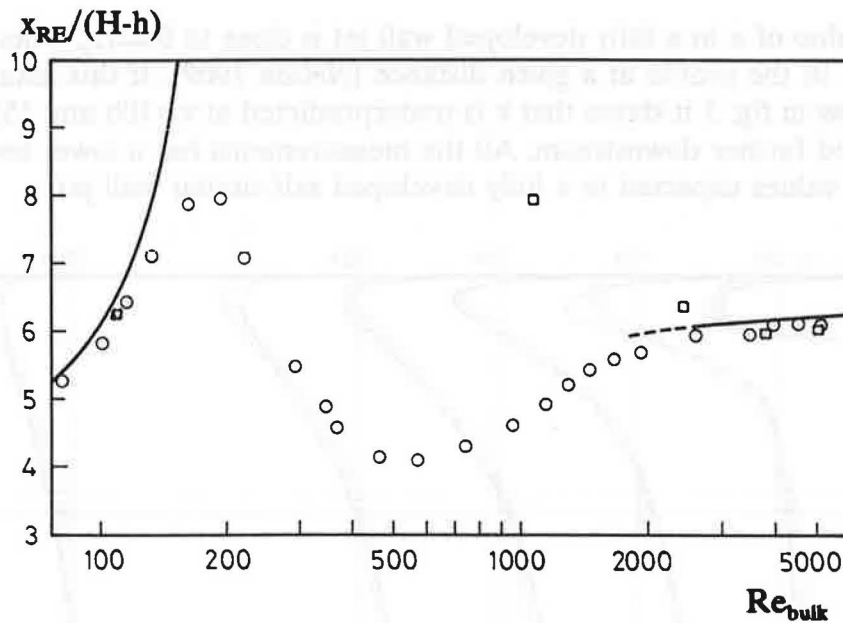

Figure 4. Recirculation length vs. Re number. (o - measured, lines / laminar and fully turbulent simulation by Restivo 1979 and squares - LRN simulation).

DISCUSSION

The performance of the LRN model in channel flow is acceptable if the necessary fine grid, which is at least 10 points in the y^+ range from 5 to 30, is applied. This again means that it can be used to predict flow close to walls or obstacles as for example to calculate heat transfer coefficients and to calculate velocity distribution in a wall jet with low Re numbers etc.

If we look at the backward facing step test we see that there is still a region in the transitional regime where the LRN model, as well as the high Re number version, fails to give converged results. The explanations of this may be many: The recorded timedependent behaviour of the flow in the transitional regime is not taken into account in the simulation which it might be required if we want to calculate transient flow (as already discussed by *Restivo 1979*). The resolution of the grid in the shear layer may have to be higher than used in present simulations where the same grid was used for all Re numbers. The reason may also be that the LRN turbulence phenomena we find in the recirculation zone has a different character than turbulence in the boundary layer, so in order to capture this, the model has to be tuned for these phenomena as well as for the near-wall behaviour of the turbulence parameters.

That LRN phenoma arising from different sources are occuring in ventilated spaces is an established fact. In order to be able to predict these some work has to be done

in the above mentioned areas so a better understanding of the different phenomenas can be obtained.

LIST OF REFERENCES

Chen, Q. Suter, P. and Moser, A.
Influence of Air Diffusion, Energy systems laboratory, series Federal Inst. of Tech. ETH, Zürich, Switzerland, *1990*.

Heiselberg, P. and Nielsen, P.V.
The Contaminant Distribution in a Ventilated Room With Different Air Terminal Devices. RoomVent '87, Sweden, *1987*.

Huang G.P.
The Computation of Elliptic Turbulent Flows with Second Moment Closure Models. Ph.D. - Thesis, UMIST, Manchester, *1986*.

Inze, N.Z. and Launder B.E.
On the computation of buoyancy - driven turbulent flows in rectangular enclosures. UMIST, Manchester, UK, *1989*.

Laufer, J.
Investigations of Turbulent Flow in a Two - Dimensional Channel, NACA report 1053, *1949*.

Launder B.E. and Sharma B.I.
Letters in Heat and Mass transfer. *1978*, 1, 129.

Launder B.E. and Tselepidakis D.P.
Directions in Second - Moment Modelling of Near - Wall Turbulence, UMIST, Manchester, UK. 29 th Aero space Sciences Mecting AIAA 91 - 0219, Nevada, *1990*.

Murakami, S., Tanaka, T. and Kato S.
Numerical Simulation of Air Flow and Gas Diffusion in Room Model - Correspondence Between Simulation and Model Experiments. University of Tokyo *1983*.

Nelson, J.L.
An experimental investigation of the turbulent and mean flow properties of a plane two-dimensional turbulent wall jet. Dissertation, University of Tennesee. Dept. of Chem. Eng. *1969*.

Nielsen, P. V. and Möller Å.T.A.
Measurements on Buoyant Jet Flows From Ceiling Mounted Slot Diffuser. III seminar on Appl. of fluid mechanics in enviromental protection, Silesion Tech. University, Gliwice, Poland, *1988*.

Patel V.C., Rodi W. and Scheurer G.

Turbulent Models for Near - Wall and Low Reynolds Number Flows: A Review AIAA Journal, Vol 23, No. 9, *1985*.

Restivo A.M.O.
Turbulent flow in ventilated rooms, Imp. Coll. of Science and Tech., Mech Eng. Dept. Ph.D. - Thesis, *1979*.

Skovgaard, M., Hyldgård, C.E. and Nielsen, P.V.
High and low Reynolds Number Measurements in a room with an Impinging isothermal Jet, Roomvent '90, Oslo, Norway, *1990*.

Skovgaard, M. and Nielsen, P.V.
Simulation of Simple Test Case, Case 2D1. IEA, Annex 20 report, *1991a*.

Skovgaard, M. and Nielsen, P.V.
Modelling complex inlet geometries in CFD - applied to air flow in ventilated rooms. 12th IAVC Annual conference, Canada, sept. *1991b*.

Tselepidakis D.P.
Private Communications, UMIST, Manchester, UK. *1991*.