Simulation of Simple Test Case 2D1

Two-Dimensional Isothermal Forced Convection

Skovgaard, M.; Nielsen, Peter Vilhelm

Publication date:
1991

Document Version
Publisher's PDF, also known as Version of record

Link to publication from Aalborg University

Citation for published version (APA):

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- Users may not further distribute the material or use it for any profit-making activity or commercial gain
- Users may freely distribute the URL identifying the publication in the public portal

Take down policy
If you believe that this document breaches copyright please contact us at vbn@aub.aau.dk providing details, and we will remove access to the work immediately and investigate your claim.

M. SKOVGAARD & P. V. NIELSEN
SIMULATION OF SIMPLE TEST CASE, CASE 2D1
(Two-Dimensional Isothermal Forced Convection)
SEPTEMBER 1991

ISSN 0902-7513 R9131

M. Skovgaard & P. V. Nielsen
Simulation of Simple Test Case, Case 2D1
(Two-Dimensional Isothermal Forced Convection)
September 1991

ISSN 0902-7513 R9131
RESEARCH ITEM 1.46dk

SIMULATION OF SIMPLE TEST CASE, CASE 2D1
(two-dimensional isothermal forced convection)

Author : Skovgaard, M. and Nielsen, P.V.
The University of Aalborg, Department of Building Technology and Structural Engineering.
Sohngårdsholmsvej 57, 9000 Aalborg, Denmark

Date : May the 1st. 1991 (revised 25.07.91).
Report no. :
Distribution : Unlimited.
Available : The University of Aalborg.

Table of contents

- List of figures
- Summary
- Acknowledgements
- Introduction
- Simulation method
  - Numerical model
  - Computer code
- Test case - geometry and boundary conditions
- Results and discussion
  - Comparisons with LDA measurements
  - Other numerical predictions
- References 3

3
3
4
5
5
9
9
9
LIST OF FIGURES

Fig. 1. The geometry of test case 2D1.
Fig. 2. The predicted velocity field U/U₀ and the distribution of turbulent kinetic energy k²/U₀.
Fig. 3. Measured and simulated flow profiles.
Fig. 4. Comparisons between predicted and measured values in cross section y = h/2 and y = H-h/2.
Fig. 5. Decay of the maximum velocity in the wall jet.
Fig. 6. The growth in wall jet width.
Fig. 7. Predictions of the velocity distribution in the two-dimensional test case.

SUMMARY

The turbulent flow pattern is calculated with a low Re number version of the k-ε model in a room with two-dimensional isothermal flow.

The results are compared both to LDA measurements obtained in a scale model and to other data obtained by numerical simulation.

The overall performance is good and indeed satisfactory. With respect to maximum velocity and turbulence level in the occupied zone the results are very good and with respect to the decay of the maximum velocity in the wall jet and the growth of jet width small discrepancies are found.

ACKNOWLEDGEMENTS

This report is partly made while one of the authors (Skovgaard, M.) stayed at UMIST, Manchester, UK.

The authors would like to send many thanks to the staff of UMIST, who made the stay very pleasant and especially we would like to thank Dr. N. Inze and Prof. B. Launder who have been very supportive and participating throughout the work at UMIST.
INTRODUCTION

This annex 20 report shows the results of the simulation of the test case given by Nielsen (1990). The test case is two-dimensional and is chosen because of the detailed experimental data available. The experimental result is obtained by the laser-Doppler anemometry technique Restivo (1979) and Nielsen et al. (1978).

SIMULATION METHOD

Mathematical model

The numerical predictions are obtained by a numerical solution of the time averaged momentum equations, the continuity equation and an Eddy viscosity concept for the turbulent Reynolds stresses.

The governing equations for two-dimensional steady, isothermal flow are

\[
\frac{\partial}{\partial x_i}(\rho U_j) = 0 \tag{1}
\]

\[
\frac{\partial}{\partial x_i}(\rho U_i U_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_i}\left(\mu \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \rho \mu \epsilon_j\right) \tag{2}
\]

For description of the Reynolds stresses several models can be adopted, see Patel et al. (1985). Here the Launder-Sharma (1974) version of the Jones-Lauder (1972) low Reynolds number k-\(\Bar{\epsilon}\) model is used (for detailed description see also Inze (1989)):

\[
\rho \mu \epsilon_j = \frac{2}{3} \delta_{ij} \rho k - \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) \tag{3}
\]

\[
\mu_t = C_\mu f_t \frac{k^2}{\Bar{\epsilon}} \; ; \; \; \; \Bar{\epsilon} = \epsilon - 2\mu \frac{\partial}{\partial x_i} \left(\frac{\partial k}{\partial x_i}\right)^2 \tag{4}
\]

\[
\frac{\partial}{\partial x_i}(\rho U_i k) = \frac{\partial}{\partial x_i}\left(\frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_i}\right) + P - \rho \epsilon \tag{5}
\]
\[
\frac{\partial}{\partial x_i}(\rho U_i \tilde{e}) = \frac{\partial}{\partial x_i}\left(\left(\mu + \frac{\mu_t}{\sigma_t}\right) \frac{\partial \tilde{e}}{\partial x_i}\right) + C_{\epsilon 1} f_1 \frac{\tilde{e}}{k} \rho P - C_{\epsilon 2} f_2 \frac{\tilde{e}^2}{k} + C_{\epsilon 3} \frac{\mu_t}{\rho} \left(\frac{\partial}{\partial x_i}\left(\frac{\partial U_i}{\partial x_j}\right)\right)^2
\]

where \( P \) is the generation rate due to turbulent shear effects

\[
P = -u_{ij} \frac{\partial U_i}{\partial x_j}
\]

The values of the turbulent model constants and functions can be seen in the table below.

<table>
<thead>
<tr>
<th></th>
<th>Fully turb. version</th>
<th>Low Re numb. version</th>
</tr>
</thead>
<tbody>
<tr>
<td>( C_\mu )</td>
<td>0.09</td>
<td>0.09</td>
</tr>
<tr>
<td>( C_{\epsilon 1} )</td>
<td>1.44</td>
<td>1.44</td>
</tr>
<tr>
<td>( C_{\epsilon 2} )</td>
<td>1.92</td>
<td>1.92</td>
</tr>
<tr>
<td>( C_{\epsilon 3} )</td>
<td>-</td>
<td>2</td>
</tr>
<tr>
<td>( \sigma_k )</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td>( \sigma_\epsilon )</td>
<td>1.3</td>
<td>1.3</td>
</tr>
<tr>
<td>( f_\mu )</td>
<td>1.0</td>
<td>\exp(-3.4/(1+R_t/50)^2)</td>
</tr>
<tr>
<td>( f_1 )</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td>( f_2 )</td>
<td>1.0</td>
<td>1-0.3\exp(-R_t^2)</td>
</tr>
</tbody>
</table>

\[
R_t = \frac{\rho k^2}{\mu \tilde{e}}
\]

**Computer code**

The two-dimensional elliptic flow solver **TEAM (Huang (1986))**, which has been extensively used and validated, was used to obtain the flow solutions. The code is using the finite volume technique employing the staggered grid layout to overcome the checkerboard phenomenon. The solution scheme used is SIMPLE and the differencing scheme is PLDS for all terms. A non-uniform grid is used in order to produce a finer grid in the boundary layer. In fact the solution is rather sensitive to grid layout if coarser grids are used, especially in the region \( 5 < y^+ < 30 \). The equations are solved line by line in an ADI iterative manner.

As expected the convergence is rather slow because of the slow diffusive process in the boundary layer and because of the fine grid required.
TEST CASE - geometry and boundary conditions

The geometry of the test case is $L/H = 3.0$, $h/H = 0.056$ and $t/H = 0.16$ (see fig.1) and the actual dimensions used in the present simulation are $H = 3.0m$, $L = 9.0m$, $h = 0.168m$ and $t = 0.48m$. As pointed out in the specification the inlet opening is rather large compared to practical diffusers (it can be mentioned that the ratio $a_0^2/H$ is equal to 0.035 in the three-dimensional test case), but this will only make it easier to use a standard $k$-$\varepsilon$ model. In the present case where a low Re number $k$-$\varepsilon$ model is used it is necessary to use fine grid in the opening because of the inlet location close to the ceiling. The grid used is $38 \times 78$.

![Fig. 1. The geometry of test case 2D1.](image)

The boundary conditions for the test case are:

**Inlet**
- $U_0 = 5000\mu/(\rho h) = 0.455$ m/s
- $V_0 = 0.0$
- $k_0 = 1.5(0.04U_0)^2$
- $\varepsilon = k_0^{15}/(h/10)$

**Outlet**
- $U_{out} = U_0h/t = 0.159$ m/s
- $dV/dx = d\varepsilon/dx = dk/dx = 0.0$

**Wall**
- $U = V = \varepsilon = k = 0.0$

RESULTS AND DISCUSSION

Comparisons with LDA measurements

Figure 2 shows the predicted velocity field and the distribution of turbulent kinetic energy.

Fig. 3 shows the comparison between the computed velocity and kinetic energy profiles. As seen the overall agreement is very good. It is seen that the inlet conditions for $k_0$ have very little effect on the turbulence level in a distance from the inlet, which again means...
that the turbulence is created by the shear effect. The $k^{1/2}$ level can be compared to the measured level of $(u^2)^{1/2}$ (Nielsen 1990). It is noted that the decay of $U_{\text{max}}$ is slightly higher than in the measurements which becomes more evident in fig. 5.

![Image of predicted velocity field and turbulent kinetic energy distribution](image)

**Fig. 2.** The predicted velocity field $U/U_0$ and the distribution of turbulent kinetic energy $k^{1/2}/U_\sigma$.

![Image of measured and simulated flow profiles](image)

**Fig. 3.** Measured and simulated flow profiles.

In fig 4. results from two cross sections in the $y$-direction are compared with measured data.
Fig. 4. Comparison between predicted and measured values in cross section $y = h/2$ and $y = H-h/2$. 
The effect from the shear layer outside the potential core of the jet is seen. It is seen that the recirculating flow recorded in the measurements in both cross sections is predicted in the simulation, although the magnitude of the counter flow is too low. The maximum velocity in the occupied zone is well predicted both with respect to location and to magnitude.

Fig. 5. Decay of the maximum velocity in the wall jet.

Fig. 6. The growth in wall jet width.

The decay of the maximum velocity and the downstream growth of the jet width are
depicted in fig 5 and 6. The velocity decay is slightly overpredicted and it can be seen that the faster decay corresponds to a faster growth in the jet width than recorded in the experiments (fig. 6). No investigation of grid dependency of the solution has been carried out so it is possible that some numerical diffusion may be present.

Other numerical predictions

![Graph showing velocity distribution comparison]

Fig. 7. Predictions of the velocity distribution in the two-dimensional test case.

Fig. 7 shows some other numerical predictions which can be used for comparisons. The measurements are made by Restivo 1978 and reported by Nielsen et al. 1978.

One set of predictions is made by the TEACH-T code and a k,ε turbulence model with wall functions (Nielsen et al. 1978). Fig. 7 also shows some old predictions made by vorticity-stream function as dependent variables and a k,ε model. The predictions are made at a Reynolds number of 7100 and the inlet conditions are given by a box method.

The predictions by Davidson and Olsson 1987 are made by a k,ε model and with a one-equation turbulence model. The one-equation model is giving a lower velocity level in the wall jet below the ceiling, see fig. 7.

REFERENCES

Davidson, L. and Olsson, E.
Calculation of Some Parabolic and Elliptic Flows Using a New One-Equation Turbulence Model. Proc. 5th. International Conference on Numerical Methods in Laminar and

Huang, P.G. and Leschziner, M.A.

Inze, N.Z. and Launder, B.E.

Jones, W.P. and Launder, B.E.

Launder, B.E. and Sharma, B.I.

Myers, G.E., Schauer, J.J. and Eustis, R.H.

Nielsen, P.V.,

Nielsen, P.V., Restivo, A. and Whitelaw, J.H.

Nielsen, P.V.

Restivo, A.

Schwarz, W.H. and Cosart, W.P.

Sigalla, A.