International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:2, No:12, 2008

An Optimized Multi-block Method for Turbulent Flows

M. Goodarzi, and P. Lashgari

***Abstract*—**A major part of the flow field involves no complicated turbulent behavior in many turbulent flows. In this research work, in order to reduce required memory and CPU time, the flow field was decomposed into several blocks, each block including its special turbulence. A two dimensional backward facing step was considered here. Four combinations of the Prandtl mixing length and standard k-ε models were implemented as well. Computer memory and CPU time consumption in addition to numerical convergence and accuracy of the obtained results were mainly investigated. Observations showed that, a suitable combination of turbulence models in different blocks led to the results with the same accuracy as the high order turbulence model for all of the blocks, in addition to the reductions in memory and CPU time consumption.

***Keywords*—**Computer memory, CPU time, Multi-block method, Turbulence modeling.

I. INTRODUCTION

MANY researchers have been interested in using structured grids for simulating the flow fields. But some flow fields have complicated geometry such that only unstructured grids can be used for numerical simulation. The simple programming of the structured grids, and low computational time and memory consumption, make it favorite to CFD (Computational Fluid Dynamics) researches. The multi-block method was introduced by Weatherill [1] for structural meshing of the complicated geometries. Hereafter, many researchers used multi-block method for simulating such a flow fields [2, 3].

To obtain a better resolution for numerical results, it is necessary to refine the numerical grid. It leads to a high memory consumption which needs a powerful computer processor. The absence of mentioned processor makes researchers to implement parallel processing procedure. This procedure was added to the multi-block method, and made a powerful method for simulating of complicated flow fields [4-6].

It is possible to say that, all of the multi-block applications could be classified into two categories; complicated geometries, and parallel processing. The governing equations were the same in all of the blocks. Although, some researchers used different equations or models in multi-grid approach [7], but no one employed the mentioned procedure in the multi-block method.

Today, researchers are interested in applying numerical methods involving low CPU time and memory consumption which are able to be run with personal computers. Therefore, they attempt to introduce new methods by the above mentioned characteristics. This article concerns with introducing a new multi-block approach using suitable turbulence modeling in each of the block in the flow field in order to reduce both the CPU time and memory consumption.

In many turbulent flows, there are regions with simple turbulent behaviors, and only a small portion of the flow field involves complicated turbulent behaviors. In such a flow, it is not necessary to employ higher order turbulence modeling in the simple behavior zones which consume higher CPU time and computer memory. The turbulent flow past a backward facing step is an example of such a flow in which only the zone after the backward facing step concerns complicated turbulent behaviors. Therefore, one can use a higher order turbulence model in the mentioned region and lower order turbulence model in the remaining zones for simulating the flow field. It will lead to the reduced CPU time and computer memory without losing the numerical results precision.

II. GOVERNING EQUATIONS AND BOUNDARY CONDITIONS

The governing equations for a steady two-dimensional incompressible turbulent flow are continuity and momentum in xy plane, using turbulence model based on eddy viscosity concept. They are as follow:

$$\frac{\partial u}{\partial x}+\frac{\partial u}{\partial y}=0.0 \tag{1}$$

$$\begin{aligned} u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}&=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\nu_{eff}\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)\\ u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}&=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\nu_{eff}\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right)\end{aligned} \tag{2}$$

In these equations u and v are the velocity components, p is the static pressure, ρ is the fluid density, and ν_{eff} is the effective viscosity. Two well-known turbulence models employed in this research are prandtl mixing length [8] and standard k-ε [9]. The details can be found in the references. Hence, they are not presented here.

At the inflow boundary, all of the flow variables have known values. Fully developed conditions could be used at the out flow boundary. The wall function [9] for standard k-ε model in addition to the no-slip condition, were used at the

M. Goodarzi, Assistant professor of Mechanical Engineering, Faculty of Engineering, Bu-Ali Sina University, Hamadan, IranAuthor (corresponding author to provide phone: +988118257410; fax: +988118257400; e-mail: mohsengood@ gmail.com).

P. Lashgari, Ms. student of Mechanical Engineering in Bu-Ali Sina University, Hamadan, Iran. (e-mail: pl1361@gmail.com).

wall boundaries. In order to implement boundary conditions at common boundaries between two adjacent blocks, two conditions should be satisfied. At first, all of flow variables should be continuous at these boundaries. Therefore, flow variables could be interpolated between adjacent grids in the two neighboring blocks. The second condition implies that volume flux should be conserved at these boundaries. The SIMPLE algorithm in collocated grid was used to solve the governing equations in all of the blocks. Hence, the method of Rhie and Chow [10] was used to make coupling between continuity and momentum equations. The mentioned coupling should be satisfied at common block boundaries to conserve the volume fluxes. This could be satisfied by the following equations based on Fig. 1, which shows an equal spacing grid for simplicity:

$$\begin{aligned} U1 &= u(M-1,J,1) - \frac{1}{2}[2p(M,J,1) - p(M-1,J,1) - p(M-2,J,1)] \\ U2 &= u(2,J,2) - \frac{1}{2}[p(3,J,2) - p(2,J,2) - 2p(1,J,2)] \\ pterm &= p(1,J,2) - p(M-1,J,1) \\ UF(J) &= \frac{1}{2}(U1+U2) + pterm \end{aligned} \quad (3)$$

where, *U1*, *U2*, and *pterm* are dummy parameters, and *UF(J)* is the volume flux through the common boundary between the two adjacent blocks. It should be noted that, pressure values on the common boundary were not the same during middle iterations for the neighboring blocks until the final convergence was achieved.

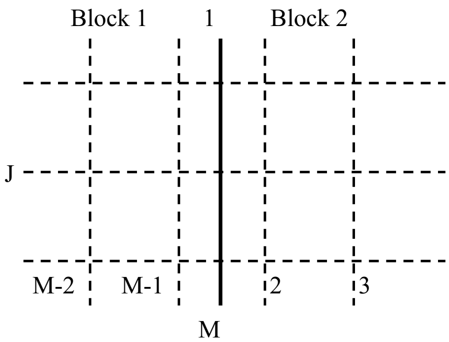

Fig. 1 Equal spaced grids near the common boundary of two adjacent blocks

The turbulent variable boundary conditions should be computed from the following equations at the common boundary, in the case of different turbulence model application in two neighboring blocks:

$$\begin{aligned} \mu_t &= \rho C_\mu \frac{k^2}{\varepsilon} \\ l_m &= C_\mu \frac{k^{3/2}}{\varepsilon} \end{aligned} \quad (4)$$

In the equation (4) k is the kinetics turbulent energy, ε is the rate of energy dissipation, l_m is the Prandtl mixing length, C_μ is a non-dimensional constant defined in the standard k-ε model, μ_t is the eddy viscosity. The Prandtl mixing length and eddy viscosity are related together in the model equations [8].

III. RESULTS AND DISSCUSIONS

In order to verify the new introduced multi-block method, turbulent flow over a backward facing step was considered. Fig. 2 shows the geometry and type of the block decomposition. The Reynolds number was 69610, the same as that in the experimental work of Kim et al [11], and computed based on the hydraulic diameter and averaged velocity in the entry region.

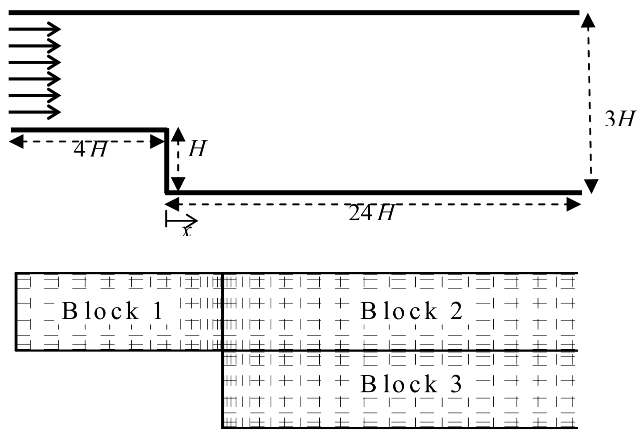

Fig. 2 Geometry, and block decomposition of the flow field

Table I shows the four different turbulence model combinations in three blocks of the flow field. The grid study was performed for the first case of these combinations and finally optimum grids were selected to be 60×80 in the first block, 200×80 in the second block, and 200×40 in the third block. These optimized grids were also used in the other combinations for making equal comparisons. The convergence criteria was selected so that to be the same for all of the mentioned combinations. This was satisfied as the residuals of the velocity components and pressure reach certain values.

Fig. 3 shows the residual of horizontal velocity component versus the number of iterations for a number of iterations. It shows that, the case number 4 which employs Prandtl mixing length model in all of the blocks, has faster rate of convergence than the other cases. As the number of blocks using the standard k-ε model increased, the rate of convergence was decreased. It should be noted that, computation time per iteration were not the same for all of the combinations. Therefore, Fig. 3 can not illustrate the whole aspects of convergence criteria. Table II shows the computed CPU time and memory consumptions for each combination. Tabulated values showed that, combinations with greater number of Prandtl mixing length model, involved lower CPU time and memory consumptions. This was an obvious result, because of the algebraic computation of the Prandtl mixing length compared to the transport equations computation of the standard k-ε model.

The two mentioned results only compared CPU time and memory consumptions. For a better comparison, the numerical results obtained by different combinations were also investigated. Fig. 4 shows the streamlines for the cases under consideration. As this figure shows, streamlines of the first three cases are closed to each other and only the fourth case is different from the others. In this figure the length of reattachment point is shorter in the last combination case compared to the three first cases. Basically, this length is a very important parameter in the separated flow fields. This length was reported by Kim et al [11] in their experimental research to be in the range of (7.0±1.0) times H, the height of step in Fig. 2. Hackman et al [12] reported this length to be 6.9 times H, in their numerical simulation. Table II shows the measured values obtained from the present numerical simulations compared to the referred experimental and numerical data.

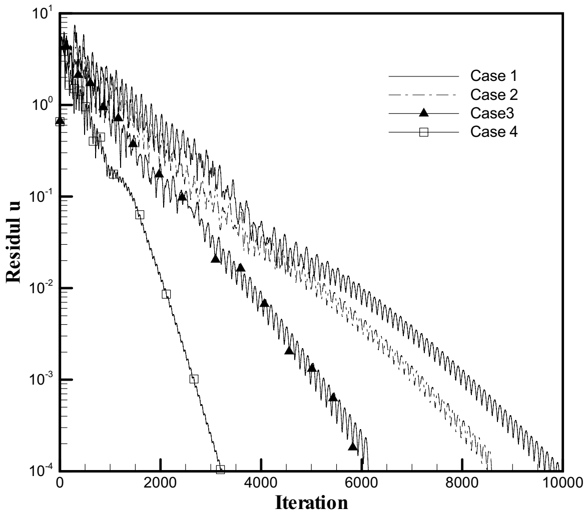

Fig. 3 Residual of the horizontal velocity component

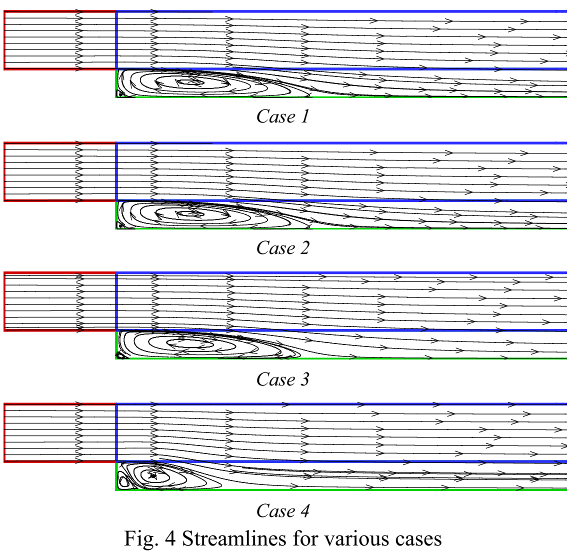

Fig. 4 Streamlines for various cases

Fig. 5 shows horizontal velocity component profiles at some sections downstream of the step, compared to the experimental data of Kim et al [11]. In all of the sections the predicted results of the two first combination cases were coincide to each other completely. They were closer to the experimental data than the other combination cases, especially case number 1. The reason was the simple turbulent behavior in the first block and more complicate behavior in the third block, in which higher order turbulence model were employed. The numerical results of the case 3 closely followed the results of cases 1 and 2. According to Table II, the standard *k-ε* model was employed only in the third block in this case. Since complicated turbulent behavior of this flow field occurred only in the third block, application of low order turbulence model in the first and second blocks did not considerably affect the accuracy of the numerical results. It should be noted that case 3 consumed lower CPU time and memory than the cases 1 and 2. Therefore, it is can be claim that by a suitable combination of turbulence modeling, one can get reasonable numerical results without large CPU time and memory consumption. The numerical results of the case 4, not only did not involve the accuracy of the three other cases, but also had different profiles compared to these cases, which closely followed the experimental data.

IV. CONCLUSION

A new multi-block approach was introduced which used suitable combination of turbulence modeling for each block. This method could predict the flow field behavior as accurate as the case of employing higher order turbulence model in all of the blocks, by implementing lower capacity of memory and computation time. Therefore, this method can be applied for simulation of complicated flows which could not be simulated by previous methods on a personal computer with a simple processor.

REFERENCES

[1] Weatherill N.P, and forsey C. R., *"Grid Genaration and Flow calculation for Complex Aircraft Geometries Using a Multi-Block Scheme"*, AIAA Paper, 85, 1985.
[2] Jae Wook Kim, and Duck Joo Lee., *"Characteristic Interface Conditions for Multiblock High-Order Computation on singular Structured Grid"*, AIAA Journal, Vol. 41, No. 12, pp.2341-2348, 2003.
[3] Epstein B, Rubin T, and Seror S., *"Accurate Multiblock Navier - Stokes Solver for Complex Aerodynamic Configurations"*, AIAA Journal, Vol. 41, No. 4, pp. 582-593, 2003.
[4] Drikakis D., *"A Parallel Multiblock Characteristic-Based Method for Three-Dimensional Incompressible Flow"*, Advances in Engineering Software, Vol. 26, pp.111-119, 1996.
[5] Di Serafino Daniela., *"A Parallel Implementation of a Multigrid multiblock Euler Solver on Distributed Memory Machines"*, parallel Computing, Vol. 23, pp. 2095-2113, 1997.
[6] Tsai H.M, Wong A.S.F, Cai J, and Liu F., *"Unsteady Flow calculations with a parallel Multiblock Moving Mesh Algorithm"*, AIAA Journal, Vol. 39, No. 6, pp.1021-1029, 2001.
[7] Atkins H.L., *"A Multi-block Multigrid Method for the Solution of the Euler and Navier - stokes Equations for Three Dimensional Flows"*, AIAA Paper, 91, 101, 1991.
[8] Prandtl L., *"Über Die Ausgebildete Turbulenz"*, ZAMM, Vol. 5, pp. 136-139, 1925.
[9] Launder B.E, and Spalding D.B., *"The Numerical Computation of Turbulent Flow"*, Comput. Methods Appl. Mech. Eng, Vol. 3, pp. 269-289, 1974.
[10] Rhie CM, Chow WL. "Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation". *AIAA J.* 1983; 21: 1525-1532.
[11] Kim J, Kline S.J and Johnston J.P., *"Investigation of a Reattaching Turbulent Shear Layer: Flow over a Backward-Facing Step"*, J. Fluid Eng., ASME Trans, Vol. 102, pp. 302-308, 1980.
[12] Hackman P.L, Raithby G.D and Strong A.B., *"Numerical Predication of Flows Over Backward Facing Step by a Finite Element Method;*

Comparison with Finite Volume Solutions and Experiments", Int. J. Numer. Methods Fluids, Vol. 4, pp. 711-724, 1984.

TABLE I
TYPES OF TURBULENCE MODEL COMBINATIONS IN VARIOUS BLOCKS

CASE NUMBER	*TURB. MODEL IN BLOCK 1*	*TURB. MODEL IN BLOCK 2*	*TURB. MODEL IN BLOCK 3*
1	Standard $k-\varepsilon$	Standard $k-\varepsilon$	Standard $k-\varepsilon$
2	Prandtl M. Length	Standard $k-\varepsilon$	Standard $k-\varepsilon$
3	Prandtl M. Length	Prandtl M. Length	Standard $k-\varepsilon$
4	Prandtl M. Length	Prandtl M. Length	Prandtl M. Length

TABLE II
CPU TIME, MEMORY CONSUMPTION, AND REATTACHMENT POINT LENGTH FOR DIFFERENTCOMBINATION CASES

Case Number	**CPU Time (min)**	**Memory Consumption (MB)**	**Reattachment point length (present)**	**Reattachment point length (Numerical [15])**	**Reattachment point length (experimental)**
1	112	125	7.2H	6.5H	(7.0±1.0)H
2	83	97	7.2H	6.5H	(7.0±1.0)H
3	70	81	6.9H	6.5H	(7.0±1.0)H
4	54	68	5.3H	6.5H	(7.0±1.0)H

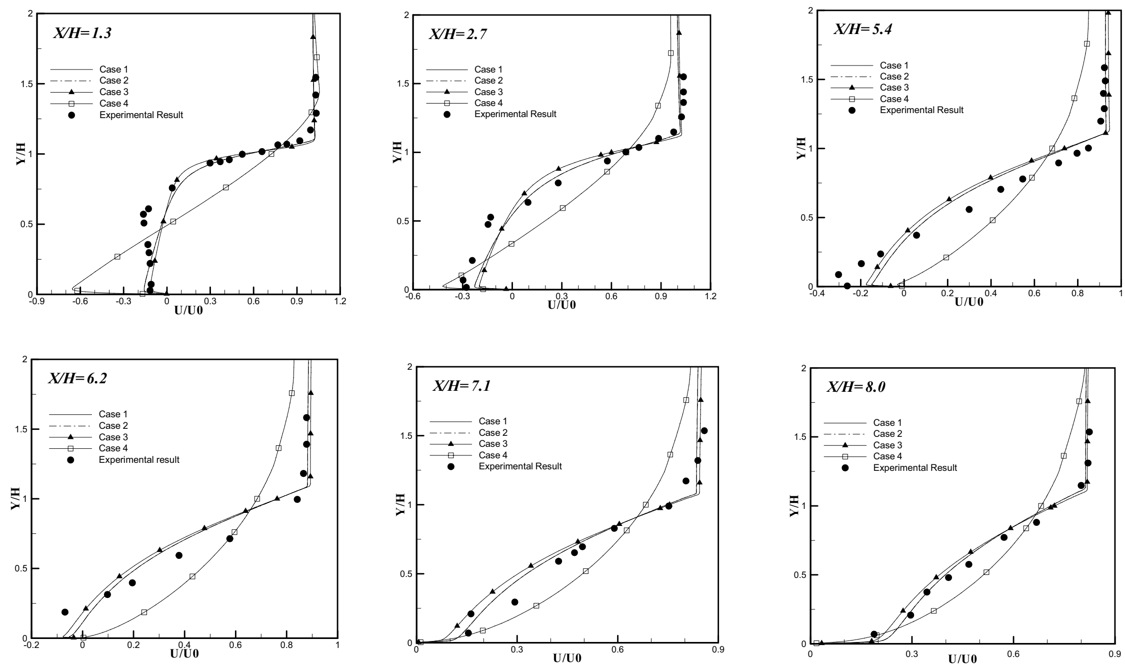

Fig. 5 Horizontal velocity profiles at several sections downstream of the step