# Turbulence Model of Compressible Flow for Non-orthogonal Grid System


## 何玟芳、謝其源

E-mail: 9612382@mail.dyu.edu.tw



**ABSTRACT**

The purpose of this study is to discuss the turbulence model of compressible flow for non-orthogonal grid system through CFD (Computational Fluid Dynamics) method. In order to simulate the complex geometry, the author uses three different wedge angles to analyze the flow field. In order to understand the physical details both in shock wave and viscous sublayer, the tip of each wedge angle is managed cautiously. And grids near the caution is also extended toward wall. The establishment of grids is extremely important for numerical convergence. This study takes finite-difference method, LU Scheme and MUSCL Scheme for the mean flow. As long as the turbulent flow, it uses central difference method, explicit and LU Decompositions to figure out the turbulent viscosity by each time step. The turbulent viscous effect is then employed to modify the original governing equation. The convergent solution is then a compressible turbulent flow solution. The flow field are 2D non-orthogonal wedge flow, and the wall function is exploited to simulate the flow within the boundary layer. The shock wave positions are predicted. Numerically, and are compared with the theoretical ones.



Keywords : compressible flow ; CFD ; non-orthogonal grid system ; turbulence effect ; turbulence model ; shock waves


**Table of Contents**



**REFERENCES**


[1] Cebeci, T., et al.,“ A General Method for Calculating Three- Dimensional Compressible Laminar and Turbulent Boundary Layer on Arbitrary Wings ”, NASA-CR-2777, 1997.

[2] Baldwin, B.S. and Lomax, H.,“ Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows ”, AIAA Paper No. 78-257.Jan, 1978.

[3] Agarwal, R.K. and Bower W.W.,“ Navier Stokes Computations of Compressible 2D Impinging Jet Flowfield Using a Two Equation Turbulent Model ”, AIAA Paper, 80-0007, 1980.

[4] Agarwal, R.K. and Bower W.W.,“ Navier Stokes Computations of Turbulent Compressible 2D Impinging Jet Flowfield ”, AIAA J., Vol.20, No.5, pp.577-584, 1982.

[5] Looney, M.K. and Walsh, J.J.,“ Mean Flow and Turbulent Characteristics of Free and Impinging Jet Flows ”, J.F.M., 147, pp.397-429, 1984.

[6] Launder, B.E. and Spalding, D.B.,“ The Numerical Computation of Turbulent Flows ”, Computer Methods in Applied Mechanics and Engineering Vol.3, pp.269-280, 1974.

[7] Launder, B.E., Prdden, C.H., and Sharna, B.I.,“ The Calculation of Turbulent Boundary Layers on Spinning and Curved Surfaces ”, ASME J., Fluid Eng, Vol.99, pp.231-239, 1997.

[8] Lin, R.S. and Chang, K.C.,“ Predictions of Free Jet Flow Field Using Various Turbulence Models ”, CSME J., Vol.10, No.3, pp.159-167, 1989.

[9] Yu, etc.“ Three-Dimensional Calculation of Supersonic Reacting Flows Using an LU Scheme ”, Sverdrup Technology, Inc. NASA Lewis Research Center Cleveland, Ohio, 1989, 7.

[10]Demirzic, I.A.,“ A finite volume method for computation of fluid flows in complex geometries ”, PhD thesis, University of London, 1982.
[11]Rhie, C.M. and Chow, W.L.,“ Numerical study of turbulent flow past an airfoil with trailing edge seperation ”, AIAA J., Vol.21, pp.1525-1532, 1983.
[12]魏大鈞,“ 壓縮流場的紊流模式研究 ”, 大葉大學碩士論文, 1998.
[13]邱聖棻,“ 壓縮流場的低雷諾數紊流模式分析 ”, 大葉大學碩士論文, 1999.
[14]Chang, K.C., Hsieh, W.D., and Chen, C.S.,“ A Modified Low- reynolds-Number Turbulence Model Applicable to Recirculation Flow in Pipe Expansion ”, ASME Journal of Fluids Engineering , Vol.117,pp.417-423,1995.
[15]朱思文,“ 低雷諾數壓縮流場的紊流模式之改進 ”, 大葉大學碩士論文, 2000.
[16]Gordon, Sandford, and McBride, Bonnie J.,“ Computer Program for the Calculation of Complex Equilibrium Compositions, Rocket Performance, Incident and Reflected Shocks, and Chapman- Jouguet Detonations ”, NASA SP-273 Interim Revision, 1976.
[17]Reid, R.C., Prausnitz, J.M., and Sherwood, T.K.,“ The Properties of Gases and Liquids ”, McGraw-Hill Publishing Co., New York, NY, 1977.
[18]Reynolds, W.C.,“ Computation of Turbulent Flows ”, 1976.
[19]Hoffmann, Klaus A., and Chiang, Steve T.,“ Computational Fluid Dynamics for Engineers-Volume Ⅱ ”, 1993.
[20]Yang, S.L.,“ Development and Application of Computational Methods in Fluid Dynamics ”, July 1995.
[21]Amsden, A.A., O ’ Rourke, P.J., and Butler, T.D.,“ Turbulent Boundary Layer Treatment in KAVA-Ⅱ ”.
[22]莊書豪、鄭東辰,“ 壁面效應對翼剖面黏性流場之影響分析 ”, 中國機械工程學會第十屆學術研討會論文集, 1993.
[23]Shapiro, A.H.,“ The Dynamics and Thermodynamics of Com- pressible Fluid Flow ”, New York, 1953.
[24]James, E.A., John,“ Gas Dynamics ”, pp.384-395, 1985.
[25]陳偉雄,“ 紊流迴流近牆區模擬傳輸過程之研究 ”, 成功大學碩士論文, 1995.
[26]陳伯欽,“ 運用非正交非交錯網格於複雜幾何形狀流場 ”, 清華大學碩士論文, 1994.
[27]蘇泓瑜,“ 運用非正交非交錯網格方法配合雷諾應力模式於複雜幾何形狀流場 ”, 清華大學碩士論文, 1996.
[28]吳文忠,“ 超音速空穴流抑制壓力震盪之數值研究 ”, 成功大學碩士論文, 1994.
[29]謝其源、溫志湧,“ 前翼尾翼組合的地效氣動力分析 ”, 交通部計畫結案報告, 1996.
[30]Lin Herng, and Chieng Ching Chang,“ Efficient Reynolds- Averaged N-S Solver with Two-Equation Model of Turbulence ”, Dissertation for Doctor of Philosophy, Graduate School of Power Mechanical Engineering, National Tsing Hua University, 1992.
[31]Chun Chuang Che, and Chieng Ching Chang,“ A Comparative Study of Higher-Order Turbulence Models on the high Speed Separated Flows ”, Dissertation for Doctor of Philosophy, Graduate School of Power Mechanical Engineering, National Tsing Hua University, 1994.
[32]Tokunaga H., Ichinose K., and Satofuka N.,“ Direct Simulation of Transient Turbulent Flows Using Generalized Method of Lines ”, FED-Vol. 117, Advances in Numerical Simulation of Turbulent Flows ASME, 1991.
[33]Kato Chisachi, and Ikegawa Masahiro,“ Large Eddy Simulation of Unsteady Turbulent Wake of a Circular Cylinder Using the Finite Element Method ”, FED-Vol.117, Advances in Numerical Simulation of Turbulent Flows ASME, 1991.
[34]Jameson Antony, and Yoon Seokkwan,“ Lower-Upper Implicit Schemes with Multiple Grids for the Euler Equations ”, Vol.25, No.7, July 1987.
[35]Jameson, A., Schmidt, W., and Turkel, E.,“ Numerical Solution of the Euler Equations by Finite Volume Methods Using Rung-Kutta Time Stepping Schemes ”, AIAA paper 81-1259, AIAA Fluid and Plasma Dynamics Conference, Palo Alto, June 1981.