第 7 卷 第 3 期　　航空动力学报　　Vol. 7 No. 3
1992 年 7 月　　Journal of Aerospace Power　　July 1992

90°强曲率弯管内三维湍流流场的数值分析

西安交通大学　　席　光　尚　虹　王尚锦

【摘要】　本文根据作者发展的一种能计算任意截面形状 90°弯管内三维不可压缩湍流流场的分析方法，利用作者实验的圆截面弯管数据（$R_c/D=0.87$）检验 k-ε 双方程湍流模型预示强曲率三维湍流流场的精度。计算了 $R_c/D=2.3$ 的矩形截面和 $R_c/D=0.87$ 的圆截面 90°弯管，并将计算结果与相应实验数据进行了比较分析。结果表明，即使对曲率很强的弯管，吸力面（内侧壁）不出现二次流迁移所形成的低速区前，标准 k-ε 双方程模型仍能较好地预示出平均速度场。

关键词：　强曲率弯管　三维湍流　数值分析

1　引　言

90°弯管在化工、换热及各种流体机械装置中应用十分广泛，研究其内部流动规律对于减小阻力损失、强化换热以及分析流体机械内部流动机理有重要意义。国外对不同曲率的弯管进行了实验和数值计算[1,2,3]。国内对不同曲率的矩形截面弯管内三维湍流流场进行了数值计算[4,5]，但对于圆截面弯管内三维湍流流场的计算尚无报道。从已有的研究结果看，弯管的曲率直径比（R_c/D）及截面形状对流动结构有重要的影响，必须进行更深入、全面的研究。特别是目前应加强对 $R_c/D<1.0$ 的极强曲率弯管的研究，这对于全面考查曲率对湍流结构的影响，改进湍流模型有重要意义。

2　计算与分析

2.1 计算方法概述

假定弯管内流场为定常、不可压缩三维湍流流场。计算采用时均 N-S 方程、k-ε 双方程湍流模型及连续性方程。计算坐标系采用能适应于不同截面形状弯管计算的（s, ξ, η）坐标系。其 s 坐标方向为弯管几何中心线的切线方向；ξ, η 坐标线为与 s 坐标方向垂直的平面上的附体坐标线，它们通过求解 2 个 2-D Laplace 偏微分方程组得到。图 1（a）、（b）为典型的 ξ, η 坐标线。

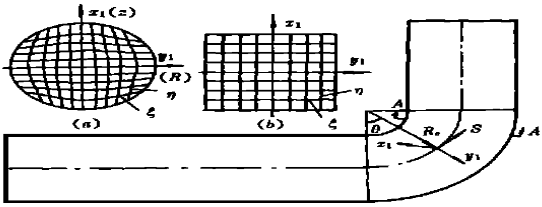

图 1　计算坐标系

本文于 1991 年 12 月收到。　＊＊ 西安交通大学风机教研室　710049

计算中对控制方程进行了半抛物化处理, 即略去了沿 s 方向的二阶扩散项。差分离散方程及边界条件的处理方法见文献 [6]。

2. 2 计算结果及分析

2. 2. 1 90 强曲率矩形截面弯管 ($R_c/D=2.3$)

矩形截面弯管选择文献 [1] 的实验弯管。为与实验条件基本一致, 计算中弯管进出口分别连接一段长为 10D 与弯管等截面的直管。本结果计算网格为 (56×11×15) 。图 2 为 θ= 60 °

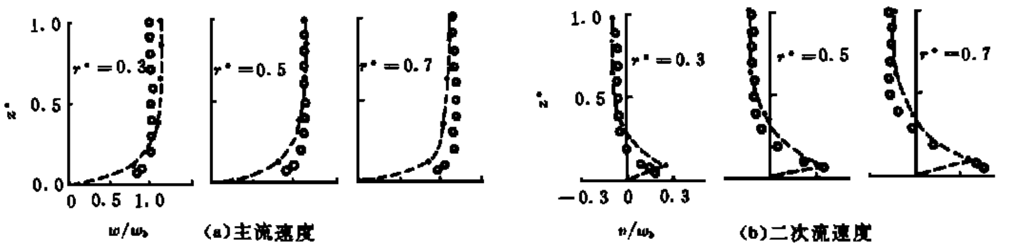

图 2 θ= 60 截面上速度分布 (a. 主流速度 b. 二次流速度)

截面上主流及半径方向二次流分布 (图中符号的定义同文献 [1]) 。可以看出计算的主流速度除壁面附近与实验值有一定误差外, 大部分区域与实验值比较接近。计算出的二次流与实验符合良好, 最大值占主流平均速度的 35% , 可见二次流较强。上述计算表明, 本文的数值分析方法和计算机程序是正确可行的。

2. 2. 2 90 强曲率圆截面弯管 ($R_c/D=0.87$)

为进一步检验本文方法计算圆截面弯管内部流动的能力, 考查标准 $k-\epsilon$ 模型预示具有极强曲率弯管内三维湍流流场的能力, 对本文作者实验测量的 $R_c/D=0.87$ 的圆截面弯管[7]进行了计算。弯管直径 $D=280$mm, 进口来流为正在发展的湍流, 来流雷诺数 $Re=2.78\times10^5$。计算中弯管进出口分别接长为 2D 的直管, 计算网格为 (20×10×10) 。图 3 为弯管进口到出口三个不同截面上平均速度分布。由图 3 (a) 可知在 0 及 30 截面上计算出的主流速度与实验符合较好, 在靠近内侧壁附近, 计算值比实验稍大。在 90 截面上计算与实验出现了较大差别, 计算没能预示出内侧壁附近的低速区 (这种低速区是由于流动方向的强逆压梯度及二次流的迁移形成的) 。从计算与实验的比较看, 计算结果能较好地反映出流动方向逆压梯度造成的内侧主流速度迅速减小的特征, 但没能预示出由于二次流

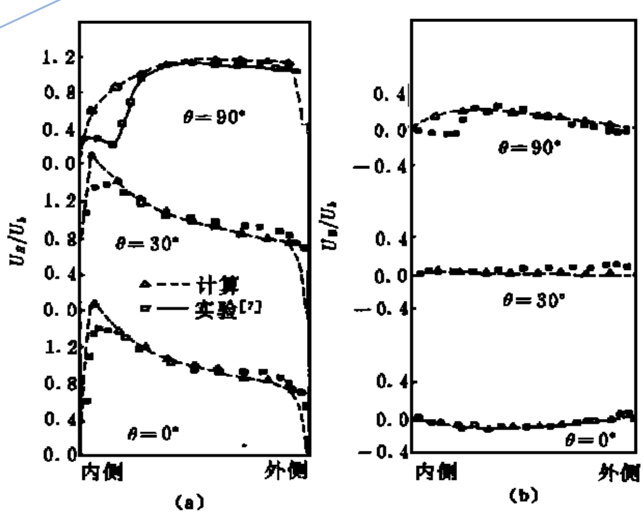

图 3 圆截面弯管不同截面平均速度分布

的迁移作用形成的平台区。这是因为弯管内侧低速平台区中, 湍流特性是高非各向同性的, k-ε 双方程模型难以精确描述这种流动。图 3 (b) 为径向方向的二次流分布。可以看出除 90 截面上内侧附近计算的二次流与实验有较大差别外, 其余区域都符合较好。

图 4 为湍流动能计算与实验的比较。由图可知, 计算出的湍流动能分布趋势与实验基本一致, 但定量上有较大差别。实验和计算都表明, 内侧壁附近的湍流动能比外侧壁大, 这是由于该截面上内侧壁处速度及速度梯度都较大, 湍流动能生成项也相应增大所造成的。此时凹壁及凸壁对湍流强度所起的加强和抑制作用已不占主导地位。

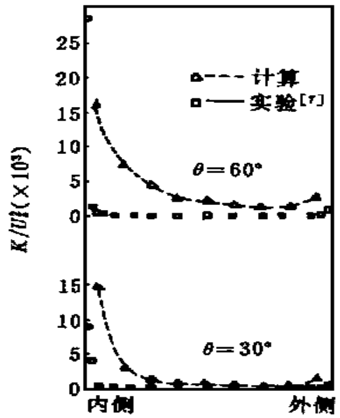

图 4　圆截面弯管湍流动能分布

3　结　论

(1) 本文发展了一种能够计算不同截面形状 90 弯管内三维湍流流场的数值分析方法。分别对矩形和圆形 90°强曲率弯管进行了计算分析, 并与有关实验数据进行了比较。计算结果表明, 本文的分析方法是切实可行的。(2) 本文的计算表明, 即使对曲率很强的弯管 (如 $R_v/D=0.87$ 的圆管), 当弯管内侧不出现由于二次流的迁移所形成的低速区前, 标准 k-ε 模型仍能较好地预示出平均速度的分布趋势。但当弯管内侧出现上述低速区后, k-ε 双方程湍流模型的预示精度大大下降。这一特点值得改进 k-ε 模型时重视。

参　考　文　献

1　Taylor A M K P, Whitelaw J H, Yianneskis M. Curved Ducts with Strong Secondary Motion: Velocity Measurements of Developing Laminar and Turbulent Flow. ASME, Jour. of Fluids Eng., 1982, 104 (3)

2　Azzola J, et al. Developing Turbulent Flow in a U-Bend of Circular Cross-Section: Measurement and Computation. ASME, Jour. of Fluids Eng., 1986, 108 (3)

3　Al-Rafai W N, et al. A Study of Turbulent Flows in Pipe Bends. IMechE, Part C: Jour of Mechanical Eng. Science, 1990, 204

4　郭荣伟. S 形管道中三元粘性气流的分析和数字解. 力学学报, 1985, 17 (4)

5　焦德勇等. 弯曲扩压管道中三维不可压粘性流的数值计算及湍流模型的研究. 应用数学和力学, 1989, 10 (7)

6　王尚锦, 席光, 苗永淼. 复杂几何边界三维粘性内流问题的计算方法及算例. 工程热物理学报, 1991, 12 (3)

7　尚虹. 90 强曲率圆截面弯管内三维紊流流场的实验及理论研究: [硕士论文]. 西安交通大学, 1991

NUMERICAL ANALYSIS OF 3-D TURBULENT FLOWS IN 90°BENDS

Xi Guang, Shang Hong, Wang Shangjin (*Xian Jiaotong University*)

ABSTRACT The 3-D turbulent flows in 90°bends with different cross-sections have been analyzed with incompressible, time-averaged N-S equation. The capability of k-ϵ turbulence model for predicting the flow field with very strong curvature has been verified by the pipe bend ($R_c/D=0.87$) experimental data made by the authors. Calculations of 3-D flow field in a 90°duct bend and a 90°pipe bend are presented. The results show that, before occurrence of the low velocity area formed mainly by the interchange of secondary flows near the inner wall, the mean velocity field predicted by the k-ϵ turbulence model is proved in reasonably good agreement with the experimental data even though the curvature of bends is very strong. But as soon as the above-mentioned low velocity area appears, the prediction accuracy of the k-ϵ turbulence model greatly drops down.

APPLICATION OF SUBMERGED VORTEX GENERATORS FOR SEPARATION CONTROL OF A SUBSONIC DIFFUSER

Chen Xiao and Jing Ping (*Nanjing Aeronautical Institute*)

ABSTRACT The effects of the main geometrical parameters of the submerged vortex generators on the diffuser performance are analysed. A combination of corotating and counter-rotating votex generators was used in the study. In the center part of the channel, four pairs of counter-rotating vortex generators were used to eliminate the flow separation. At the two corners, two corotating vortex generator blades and one pair of counter-roataing blades were used respectively to improve the three-dimensional separation flow fields in these corner regions. The experimental results show that the static pressure recovery coefficient C_P is enhanced remarkably and the total pressure loss is also reduced, if the geometrical parameters and the location of the vortex generators are selected properly. A comparative study between the submerged vortex generators of concave shape and the conventional vane-type vortex generators is also presented. Both of them have almost the same efficiency for separation control, but the parasitic drag of the former is less due to its submergence in the boundary layer; and it is convenient for practical application because its distance to the separation starting point on the diverging wall of the generator is much shorter.

TURBOFAN WITH A SMALL BYPASS RATIO FOR FIRST STAGE OF A TWO-STAGE TO-ORBIT VEHICLE

Yang Kaitian (*Shenyang Aeroengine Research Institute*)

ABSTRACT As turbomachinary is superior to rocket and ramjet at lower flight speed, a computational analysis of five sorts of hydrogen-fueled turbomachinary, i. e. ATR engine, rocket fan, tip turbine fan, two-spool hydrogen expander engine and turbofan has been accomplished. The results show that the turbofan with small bypass ratio is superior to the others in the performance at flight Mach number below 3.5. After choice of better cycle parameters, further performance calculation of the turbofan has been carried out. The total residual thrust of six engines is shown to be more than 411.88kN at flight Mach number below 3.5. It is indicated that the specific impulse of the engine is more than 31.63kN · s/kg at flight mach number below 3.5 and 73.55kN · s/kg at flight Mach number below 0.9 respectively. This type of turbofan combined with a ramjet is proved to be suitable for a powerplant