Numerical Simulation of Flows past a Circular and a Square Cylinder at High Reynolds Number, and a Curved Plate in Transitional Flow

Ling Zhang
Washington University in St. Louis

Follow this and additional works at: https://openscholarship.wustl.edu/eng_etds

Part of the Mechanical Engineering Commons

Recommended Citation
https://openscholarship.wustl.edu/eng_etds/264
Numerical Simulation of Flows past a
Circular and a Square Cylinder at High Reynolds Number,
and a Curved Plate in Transitional Flow
by
Ling Zhang

A thesis presented to
the School of Engineering and Applied Science
of Washington University in
partial fulfillment of the
requirements for the degree
of Master of Science

May 2017
St. Louis, Missouri
# Table of Contents

List of Figures .................................................................................................................. iv
List of Tables ..................................................................................................................... vii
Acknowledgments ........................................................................................................... viii
Abstract ......................................................................................................................... ix

Chapter 1: Introduction ..................................................................................................... 1
  1.1 Motivation ................................................................................................................. 1
  1.2 Background ............................................................................................................... 2
    1.2.1 Circular and Square Cylinder ........................................................................... 3
    1.2.2 Curved Plate ....................................................................................................... 5
  1.3 Goals and Objectives ............................................................................................... 6

Chapter 2: Computational Method, Mesh & Boundary Conditions ................................. 8
  2.1 Circular Cylinder ...................................................................................................... 8
    2.1.1 Grid Generation ................................................................................................. 8
    2.1.2 Boundary Conditions ....................................................................................... 10
  2.2 Square Cylinder ...................................................................................................... 10
    2.2.1 Grid Generation ................................................................................................. 10
    2.2.2 Boundary Conditions ....................................................................................... 12
  2.3 Curved Plate ............................................................................................................ 12
    2.3.1 Grid Generation ................................................................................................. 12
    2.3.2 Boundary Conditions ....................................................................................... 14

Chapter 3: Simulations Results and Discussion ................................................................. 16
  3.1 Flow past a Circular Cylinder .................................................................................... 16
    3.1.1 Pressure Coefficients and Drag Coefficients .................................................. 16
    3.1.2 Contours from Numerical Simulations ............................................................. 23
  3.2 Flow past a Square cylinder ...................................................................................... 29
    3.2.1 Pressure Coefficients from Numerical Simulations ......................................... 29
    3.2.2 Contours from Numerical Simulations ............................................................. 31
  3.3 Flow past a Curved Plate .......................................................................................... 33
    3.3.1 Drag and Lift Coefficients from Experiment .................................................... 33
    3.3.2 Lift and Drag Coefficients from Numerical Simulations ................................... 34
List of Figures

Figure 2.1 Computational domain of circular cylinder .......................................................... 9
Figure 2.2 Mesh layout for circular cylinder simulation .......................................................... 10
Figure 2.3 Computational domain around the square cylinder .............................................. 11
Figure 2.4 Computational grid and geometry for flow past a square cylinder ..................... 12
Figure 2.5 Geometry of curved plate ...................................................................................... 13
Figure 2.6 Computational domain around the Curved plate .................................................. 13
Figure 2.7 Computational geometry and grid for flow past a curved plate ......................... 14
Figure 3.1 Time averaged pressure coefficient from leading edge to trailing edge of the circular cylinder at Re = 3.6×10^6. ................................................................. 17
Figure 3.2 Time averaged skin friction coefficient from leading edge to trailing edge of cylinder at Re = 3.6×10^6. ........................................................................... 18
Figure 3.3 Time averaged pressure coefficient from leading edge to trailing edge of the circular cylinder at Re = 1×10^6. ................................................................. 19
Figure 3.4 Time averaged skin friction coefficient from leading edge to trailing edge of cylinder at Re = 1×10^6. ........................................................................... 20
Figure 3.5 Time averaged pressure coefficient from leading edge to trailing edge of cylinder at Re = 6.7×10^5. ................................................................. 21
Figure 3.6 Time averaged skin friction coefficient from leading edge to trailing edge of cylinder at Re = 6.7×10^5. ................................................................. 22
Figure 3.7 Time averaged velocity contours averaged over 20 cycles ......................... 25
Figure 3.8 Time averaged pressure contours averaged over 20 cycles ......................... 27
Figure 3.9 Streamlines at one instant of time during simulation ........................................ 29
Figure 3.10 Time averaged pressure coefficient from leading edge to trailing edge of square cylinder at Re = 22000 ................................................................. 30
Figure 3.11 Time averaged velocity contours (averaged over 20 cycles) at Re = 22000 .... 31
Figure 3.12 Time averaged pressure contours (averaged over 20 cycles) at Re = 22000. .... 32
Figure 3.13 Streamlines at one instant of time at Re = 22000.......................................................... 33

Figure 3.14 Lift coefficient CL (scale on the left axis) versus Reynolds number and drag coefficient CD (scale on the right axis) versus Reynolds number from the experiment [20]. .................................................................................................................. 34

Figure 3.15 Computed time averaged lift coefficient versus Reynolds number compared to the experiment lift coefficient versus Reynolds number .............................................................................. 36

Figure 3.16 Computed time averaged drag coefficient versus Reynolds number compared to the experimental drag coefficient versus Reynolds number ......................................................... 37

Figure 3.17 Velocity contours from different models and corresponding velocity profile at Reynolds number 150,000. ......................................................................................................................... 38

Figure 3.18 Velocity contours from different models and corresponding velocity profile at Reynolds number 180,000. ......................................................................................................................... 39

Figure 3.19 Velocity contours from different models and corresponding velocity profile at Reynolds number 220,000. ......................................................................................................................... 40

Figure 3.20 Velocity contours from different models and corresponding velocity profile at Reynolds number 250,000. ......................................................................................................................... 41

Figure 3.21 Velocity contours from different models and corresponding velocity profile at Reynolds number 400,000. ......................................................................................................................... 41

Figure 3.22 Velocity contours from different models and corresponding velocity profile at Reynolds number 600,000. ......................................................................................................................... 42

Figure 3.23 Velocity profile at x/c = 1.2 at different Reynolds number compared with experimental data .................................................................................................................................................. 44

Figure 3.24 Velocity profiles at x/c = 1.5 for different Reynolds number compared to the experimental data .............................................................................................................................................. 44
List of Tables

Table 3.1 Computed and measured Cd for flow past a circular cylinder at various Re (“~” denotes unavailable data) ........................................................................................................................................ 23

Table 3.2 Computed and experimental Cd for flow past a square cylinder (“~” denotes unavailable data). ........................................................................................................................................ 30
Acknowledgments

First of all, I would like to express my deep gratitude to my advisor Prof. Agarwal for his outstanding guidance during the course of this research. It would not be possible to complete this research without his continuous help, great patience and understanding.

Special thanks go to Tim Wray and Xu Han for their help in implementation of the Wray-Agarwal turbulence model in the CFD solver ANSYS Fluent and for providing many suggestions during the course of this research.

Thanks to Prof. Peters and Prof. Karunamoorthy for serving on the thesis committee. I would also like to thank Prof. Qiulin Qu for his encouragement and insightful comments.

Thanks to all members of the CFD lab for their support and encouragement.

Ling Zhang

Washington University in St. Louis

May 2017
Dedicated to my parents

Xiaolong Zhang and Hong Ling for their unconditional love and support
ABSTRACT OF THE THESIS

Numerical Simulation of Flows Past a Circular and a Square Cylinder at High Reynolds Number, and Curved Plate in Transitional Flow

by

Ling Zhang

Master of Science in Mechanical Engineering
Washington University in St. Louis, 2017

Research Advisor: Professor Ramesh K Agarwal

Increasing the prediction accuracy and computational efficiency of turbulence models at high Reynolds number remains a challenging problem in Computational Fluid Dynamics (CFD). In this paper, several turbulence models are applied for numerical simulation of flow past a circular and a square cylinder at high Reynolds number. Wray-Agarwal (WA) turbulence model is a recently developed one-equation turbulence model derived from k-ω closure. Comparisons are made among computational results from WA model, Spalart-Allmaras (SA) model, the shear stress transport SST k-ω model and the standard Wilcox k-ω model. For circular cylinder, the computations are performed for Reynolds numbers $Re = 6.7 \times 10^5, 1 \times 10^6$ and $3.6 \times 10^6$ and simulation for a square cylinder is performed at a Reynolds number $Re = 2.2 \times 10^4$. The computed results are assessed against previous simulations and experimental measurements. Both circular and square geometries produce vortex wakes and oscillating lift and drag. According to the results, the new WA model is competitive in accuracy with the two-equation models and has computational efficiency of a one-equation model. Another case of transitional flow past a circular arc is simulated in this thesis. For this case it has been found experimentally that a sharp and
sudden increase in lift and decrease in drag occurs at a certain Reynolds number (called the lift and drag crisis). The flow is computed using the Transition SST model, Transition k-k_l-ω model and SST k-ω model as well as a laminar flow model for Reynolds numbers slightly below and higher than $2 \times 10^5$ at which the sharp and sudden increase in both lift and drag is observed. Computations show that the transition models provide results closer to the experimental data. When flow changes from laminar to turbulent close to the critical Reynolds number of $2 \times 10^5$, the laminar-turbulent transition is responsible for sudden rise in lift and drag.
Chapter 1: Introduction

1.1 Motivation
As the computational power of computers has grown by orders of magnitude since 1960’s, Computational Fluid Dynamics (CFD) has become an efficient and cost effective method for predicting the complex flow fields around and in 3D geometries of a variety of industrial products. It is regularly employed in aerospace, automobile and ship industries for prediction and optimization of the aerodynamic and hydrodynamic performance of air, ground and marine vehicles. However, the prediction of turbulent flow fields at high Reynolds number using Reynolds-Averaged Navier-Stokes (RANS) equations with turbulence models remains a challenging task especially for separated and unsteady flows. In spite of the simple geometry of circular, square and arc cylinder, it remains very challenging to simulate their flow fields accurately because of flow separation, flow transition, unsteadiness and the shedding of vortices in the wake. The two standard test cases of flow past a circular and square cylinder correspond to non-fixed separation (separation location depends on the Reynolds number) and fixed separation (separation location does not depend on the Reynolds number) respectively. Unsteady turbulent separation plays a significant role in the flow behavior of these flows at high Reynolds number.

2D Unsteady Reynolds Averaged Navier-Stokes (URANS) equations have been widely used in simulation of these flows using a variety of well-known turbulence models, namely the one-equation Spalart-Allmaras (SA) model, and the two-equation SST k-ω model and the standard Wilcox k-ω model. Recently, a one-equation model known as the Wray-Agarwal (WA) turbulence model [1] has been developed which has been demonstrated to be competitive in accuracy with two-equation models such as the SST k-ω model and has the efficiency of a one-equation model.
e.g. the SA model. In this paper, SA, standard Wilcox k-ω, SST k-ω, and WA models are used in conjunction with the URANS equations to compute the flow field of flow past a circular and a square cylinder at high Reynolds numbers.

The flow past a circular cylinder is computed at \( \text{Re} = 6.7 \times 10^5 \) and \( \text{Re} = 3.6 \times 10^6 \). It is known that transitional flow occurs which causes reduction in drag coefficient at critical Reynolds numbers in the range \( \text{Re} = 2.5 \times 10^5 \sim 3.5 \times 10^5 \). Both the Reynolds numbers chosen for simulation are in fully turbulent regime. In the literature, the flow past a square cylinder has received much less attention compared to the circular cylinder. The experiment for flow past a square cylinder was performed by Lyn et al. [2, 3] at Reynolds number \( \text{Re} = 21400 \) using Laser-Doppler Velocimetry (LDV). Numerical simulations for this case have been performed with 2D URANS and 3D large eddy simulation (LES) and also with Detached-Eddy Simulation (DES). In 1999, LES was used by Murakami et al. [4]; they compared the results from 2D URANS and 3D LES and found that LES provided better predictions. In this paper, 2D URANS computations are performed at Reynolds number of 22000 using the WA and SST k-ω models.

1.2 Background
External flow around objects has been a topic of research for over a hundred years and it remains challenging for researchers even today. It is encountered in many industrial applications e.g. airfoils have streamline shapes in order to increase the lift and reduce the drag exerted by the external flow. On the other hand, flow past a blunt body, such as a circular and square cylinder has relevance in many applications such as wind over power lines and bridges. Periodic vortex shedding behind a cylinder can be dangerous e.g. it led to collapse of Tacoma Narrows Bridge in 1940.
1.2.1 Circular and Square Cylinder
For flow past cylinders, boundary layer separation and flow oscillations in the wake region behind the body occur due to shedding of vortices at moderate to high Reynolds numbers. At low Reynolds number, the flow is laminar and the leeside vortices remain attached to the object. As the Reynolds number increases, the flow changes from laminar to transitional to turbulent. Flow transition is a highly complex process; it has not been fully understood to date. After the critical Reynolds number range, flow becomes fully turbulence. In certain Reynolds number range, periodic shedding of vortices occurs due to unsteady separation known as the Karman vortex street. This periodic shedding of vortices from either side of the cylinder creates an oscillatory flow at a discrete frequency, specific to the Reynolds number of the flows. The flow phenomena such as boundary layer separation, vortex shedding, transition and turbulence are very common in flow over an aircraft at high angle of attack. By simulating flow past simple geometries such as a circular and a square cylinder, a great deal of insight in the flow field can be obtained which is relevant to flow over an aircraft.

There are many experimental studies in the literature that provide the measurements of pressure coefficients, skin friction coefficients and drag coefficients of circular cylinder at various Reynolds numbers. However, when Reynolds number is higher than the critical Reynolds number, the results from various experiments shows little agreement; it seems that the results are very sensitive to small changes in the experimental conditions e.g. the turbulence intensity at high Reynolds number. In 1953, Delany and Sorensen [21] provided the measurements in the Reynolds number range $10^6 \sim 2 \times 10^6$. In 1970, Roshko [6] measured the pressure coefficients and drag coefficients at high Reynolds number from $10^6$ to $10^7$. He performed the experiment in the subsonic test section of the Southern California Cooperative Wind Tunnel (CWT). The results showed that the
drag coefficient increased from 0.3 to 0.7 in the range of Reynolds number $10^6 \sim 3.5 \times 10^6$. When Reynolds number increased beyond $3.5 \times 10^6$, the drag coefficient became nearly constant at 0.7, indicating that the flow was no longer transitional and became fully turbulent.

With increase in computational power in the past several decades, computational fluid dynamics (CFD) can now be used to obtain accurate predictions of unsteady turbulent flow in the high Reynolds number range. However, difficulties remain in applying CFD in case of flow past a circular cylinder. Numerical simulations for flow past a circular cylinder have been conducted at high Reynolds numbers by Andrew et al. [9] in 2015. Their simulation results show that Reynolds-averaged Navier Stokes (RANS) equations and Unsteady Reynolds Averaged Navier Stokes (URANS) equations with turbulence models provide results with similar trends. Menter’s Shear Stress Transport (SST) k-\omega turbulence model captures the flow properties with higher accuracy than other two-equation turbulence models. Another simulation by Pietro et al. [5] applied large-eddy simulation (LES) at Reynolds number of $5 \times 10^5$ and $10^6$. They found that results from LES had better agreement with the experimental data compared to the RANS results. His results agreed reasonably well with the experimental data for velocity distributions and streamwise Reynolds stresses.

Flow past a square cylinder, however, has received less attention compared to that for flow past a circular cylinder. Both circular and square cylinder have simple geometry, and there is a relationship between these two cases. The circular cylinder has a non-fixed boundary layer separation; however, square cylinder has a fixed separation from downstream corners. Features that are less apparent in circular cylinder case could be more distinct in the case of square cylinder.
To observe the turbulent wake behind a square cylinder at Reynolds number of 21400, Lyn et al. [2] used a two-component Laser Doppler Velocimeter (LDV). Mean streamwise velocity along the center line, streamlines, contours of the turbulent stresses etc. were analyzed in detail. The results showed some similarities with the previous measurements of flow past a circular cylinder. The results also showed that the flow length scales were larger compared to the circular cylinder in the near wake and in the streamwise direction.

Flow past a square cylinder has been simulated by Lo et al. [14] at Reynolds number of 22000 with LES and DES models. Both models provide prediction of pressure coefficients that agree with the experimental data. However, the geometry of the sharp corner of the cylinder causes a sharp turn in the pressure distribution in CFD simulation. The streamwise velocity and drag coefficient provided by LES and DES are in agreement with the experimental data. In 2015, Sercan et al. [22] conducted transient unsteady simulations employing LES at the Reynolds number of 5000 and 10000. Comparisons of time-averaged streamlines and velocity contours were made with experimental results to analyze the patterns. Yong et al. [15] also employed by LES to conduct simulations at Reynolds number of 22000. In this case, the grid at the corners of the square was smoothened by providing a small curvature at corners to minimize the numerical error.

Results of time-averaged pressure distributions at the corners agree well with the experiments for all cases tested in the present study.

1.2.2 Curved Plate
Circular and square cylinder are symmetrical objects. A thin curved plate is an asymmetrical object. A recent experiment by Patrick et al., which focus on nonsymmetrical obstacles showed interesting phenomenon. A sharp transition in lift coefficient was observed simultaneously with the drag crisis at transitional Reynolds numbers. Drag crisis is well known although not completely understood.
There are a variety of factors that can influence the drag crisis such as the geometry and surface roughness of the object. There are many applications that include asymmetrical objects such as aircraft wings, propellers, compressors, fans, and turbines. To achieve high lift and low drag, airfoil/wings usually have streamlined shapes. Most commercial airplane wings have asymmetric airfoil sections since they can generate lift at zero angle of attack. On the other hand, a symmetric airfoil is better suited for an inverted flight of an aerobatic airplane.

In the experiment of Bot et al. [20], a two-dimensional high-camber plate was placed in a hydrodynamic tunnel to measure the forces and the velocity fields. Velocity was adjusted to achieve different Reynolds number. A lift crisis was observed in the drag crisis Reynolds number range in the experiment. The critical Reynolds number was $Re = 2 \times 10^5$. In the transition range, lift coefficient increased from -3 to 8.5, while the drag coefficient dropped from 0.22 to 0.13 and the separation point location on the upper surface of the plate showed a sharp increase towards the downstream side. The lift crisis was also found in other nonsymmetrical objects. It is an extremely complex phenomenon; it is investigated in this thesis by numerical simulation.

### 1.3 Goals and Objectives
The goal of this thesis is to apply the CFD technology for flow past blunt bodies at high Reynolds number to assess the accuracy of various turbulence models, transition models and the new Wray-Agarwal (WA) model. The two benchmark cases of flow past a circular cylinder and a square cylinder, corresponding to non-fixed flow separation and fixed flow separation respectively are computed. Results from one-equation and two-equation turbulence models are compared with the results from the one-equation WA model and experimental data. Another numerical simulation is conducted for flow past a high-camber plate at high Reynolds number. Recent experiment has shown that a lift crisis with sharp jump in lift can be observed along with the drag crisis in the
critical transitional Reynolds number range. The sharp increase in lift has not been simulated in the literature before. Simulations were carried out by applying the transition models to study the flow on this nonsymmetrical object.
Chapter 2: Computational Method, Mesh & Boundary Conditions

2D URANS computations are performed using the commercial CFD solver ANSYS FLUENT. Pressure-based solver is employed for the solution of URANS equations in a finite-volume framework. Second-order discretization is used for both the convection and viscous terms. SIMPLE algorithm is used to ensure pressure/velocity coupling. Spalart–Allmaras (SA), standard Wilcoxon $k-\omega$ and SST $k-\omega$ turbulence models are built into the FLUENT. A User Defined Function (UDF) is written for the WA model. Mesh and boundary conditions used for circular cylinder and square cylinder are describe below. It should be noted that the mesh independence study for all computed solutions was conducted. Only the results from the final selected mesh are presented.

2.1 Circular Cylinder

2.1.1 Grid Generation
Calculations for flow past a circular cylinder are based on the experiment of Roshko [6]. Velocity and geometry were adjusted to obtain the Reynolds number of $6.7 \times 10^5$, $1 \times 10^6$ and $3.6 \times 10^6$. A larger computational domain is employed to minimize the influence of walls. Figure 1 shows the computational domain around the circular cylinder. The radius of the C boundary is 50D and the distance between the center of the cylinder and the outlet boundary is 75D.
Figure 2.1 Computational domain of circular cylinder.

Two-dimensional structured mesh around the circular cylinder is built with ICEM. The number of total elements in the mesh is 64800. The diameter of cylinder is 2 meters for Reynolds number of $\text{Re} = 3.6 \times 10^6$, 0.25 meters for Reynolds number of $\text{Re} = 6.7 \times 10^5$ and $\text{Re} = 1 \times 10^6$.

Both O-grid and C-grid are used in generating a suitable mesh around an airfoil. Since C-grid is aligned with the wake at the trailing edge of the airfoil, it is preferred in turbulent flow calculations. Thus, in the circular cylinder case a C-grid is employed to capture the flow features in the wake of the cylinder. C-grid in the far field is shown in Figure 2.2(a). O-grid is generated to wrap around the geometry as shown in Figure 2.2(b). C and O mesh are merged in a seamless fashion as shown in Figure 2.2(a). The first cell height near the cylinder geometry is $1.5 \times 10^{-6}$ meter to ensure that $y^+ < 1$. 


2.1.2 Boundary Conditions
To match the experimental condition, \( \text{Re} = 3.6 \times 10^6 \) is matched with \( U_\infty = 22.04 \text{m/s} \), \( \text{Re} = 6.7 \times 10^5 \) is matched with \( U_\infty = 32.82 \text{ m/s} \), and \( \text{Re} = 1 \times 10^6 \) is matched with \( U_\infty = 48.98 \text{ m/s} \).

Kinematic viscosity in the simulation is calculated based on Reynolds numbers and free stream velocity for given diameter of the cylinder. Inflow velocity boundary condition is applied on the arc of the C-mesh in horizontal direction. Pressure outlet boundary condition is applied on the downstream boundary of the computational domain. Time step is set at \( 2 \times 10^{-5} \text{s} \). The dimensionless first cell height near the wall (\( y^+ \)) is less than 1.2. The far field is at a large enough distance for accurate simulation. To avoid reverse flow, boundary conditions on the upper and lower horizontal parts of C-mesh are free-stream velocity.

2.2 Square Cylinder

2.2.1 Grid Generation
Figure 2.3 shows the computational domain around the square cylinder. It is a square domain with inlet boundary at a distance of 30L and outlet boundary at a distance of 40L from the cylinder, L being the side of the square cylinder. Upper and lower boundaries are at a distance of 7L from the center of the cylinder.
Figure 2.3 Computational domain around the square cylinder.

Figure 2.4 shows the structured mesh around the cylinder. The mesh contains around 174,000 elements. The mesh is graded in the boundary layer region. This mesh was found to be sufficient to obtain mesh independent solution. The side of the square is $L = 0.00668$ meters. To minimize the numerical error due to sharp corners, a slightly curved corner geometry is created as shown in Figure 2.4. The radius of curvature of the corner is $L / 100$. The goal of rounding the corner is to minimize the numerical error when approximating the elements of the metric tensor by using the central differencing scheme. The radius if curvature is small enough and thus has no significant effect on the flow characteristics. From the simulation results shown later in this thesis, it can be seen that the prediction of separation location shows a distinct improvement with the rounded corner mesh.
2.2.2 Boundary Conditions

Flow Reynolds number is 22000 which corresponds to a free-stream velocity of 3.3059 m/s. Velocity boundary condition is applied at the inlet of the computational domain and pressure outlet boundary condition is applied at the outlet of the computational domain. Free stream velocity boundary condition is applied on the two horizontal boundaries. No slip boundary condition is used on the cylinder surface. Time step is set at $1 \times 10^{-5}$s.

2.3 Curved Plate

Numerical simulations for flow past a curved plate correspond to the most recent experiment of Bot et al. [20]. In the transition range of Reynolds number, both lift crisis and drag crisis are observed simultaneously. Numerical simulations are performed to model this interesting phenomenon.
2.3.1 Grid Generation
The geometry of the arc cylinder corresponds to the experiment of Bot et al. [20]. Figure 2.5 shows the geometry of sectional area of the curved plate. The curved plate section is a 3-mm-thick arc with radius of it is 50mm. The chord length is $c = 74.5\text{mm}$ and $t = 16.6\text{mm}$ with $t/c = 0.2228$. Thus, the curved plate has a large camber.

Bot et al.’s experiment was carried out in the IRENav hydrodynamic tunnel, with test cross-section of $192 \times 192\text{mm}^2$ and length of 1m. Considering the geometry of the tunnel, two-dimensional structured mesh was built with ICEM. The computational domain is 1.2m long and is shown in Figure 2.6.
The structured mesh around the arc cylinder in test section of tunnel is shown in Figure 2.7. The mesh contains 233,834 elements. Two of the sharp corners were slightly smoothed with the radius of curvature of the corner=\(c/100\). The first cell height near the upper and lower wall of the curves plate is \(1\times10^{-6}\) meter to ensure that \(y^+ < 1\). Far field boundary is at a distance of 1.2 m and the mesh along the centerline was refined to accurately calculate for the wake downstream.

![Mesh in the entire computational domain of the curved plate](image)

![Zoomed-in-view of mesh near the cylinder](image)

![Zoomed-in-view of mesh around the arc corner](image)

Figure 2.7 Computational geometry and grid for flow past a curved plate.

### 2.3.2 Boundary Conditions

The boundary conditions applied in the numerical simulation correspond to the experiment of carried out in the hydrodynamic tunnel. The density and viscosity of fluid are those of Bot et al. [20] liquid water at sea level. The left boundary of the domain is set as the velocity inlet in horizontal direction corresponding to the uniform velocity upstream in the water tunnel. The velocity is adjusted between 2.02 and 8.09 m/s to adjust the Reynolds number in the range
$1.5 \times 10^5$ to $6 \times 10^5$. The exit of the tunnel is set as pressure outlet. The turbulence intensity is set at $1.8\%$ at both the velocity inlet and the pressure outlet. No slip condition is applied on the surface of the curved plate.
Chapter 3: Simulation Results and Discussion

3.1 Flow past a Circular Cylinder

3.1.1 Pressure Coefficients and Drag Coefficients

URANS calculations provide results that oscillate in time. The criterion used for convergence of the solution is that the drag coefficient oscillates periodically for more than 50 time periods. Time averaged pressure coefficient and skin friction coefficient from leading edge to the trailing edge of the cylinder during 20 cycles of the converged solution are calculated. The experimental data is from Roshko et al., Jones et al. and Achenbach et al. is used for comparison. Computed results using different turbulence models such as SST k-\(\omega\), the standard Wilcox k-\(\omega\), SA and WA model predict pressure coefficients fairly close to the experimental data at \(\text{Re} = 3.6 \times 10^6\) as shown in Figure 3.1. No simulation provides satisfactory result for skin friction coefficient at this Reynolds number as shown in Figure 3.2. Large-eddy simulation (LES) and URANS models were employed by Catalano et al. for simulations at Reynolds number \(\text{Re} = 5 \times 10^6\) and \(\text{Re} = 10^6\) respectively. Their results for skin friction coefficient are also much larger than the experimental data. The same situation was found when Detached-Eddy Simulation (DES) was applied. The present results for skin friction using several turbulence models are also larger compared to the experimental data from Achenbach et al. [8] as shown in Figure 3.2. The SST k-\(\omega\) model gives results that are somewhat closer to the experimental data compared to those obtained from SA and WA model. In Figures 3.1 and 3.2, present computations are also compared with those of Andrew et al. [9] obtained with standard Wilcox k-\(\omega\) and SST k-\(\omega\) models. The two sets of computations are closer to each other. In Figure 3.5, present computations for average pressure at \(\text{Re} = 6.7 \times 10^5\) using the
standard Wilcox k-ω, SST k-ω, SA and WA model are compared with the experimental data of Pietro et al. [5], acceptable agreement is obtained; however, none of the models can compute the pressure satisfactorily in the wake region of the cylinder which remains a very challenging problem in CFD. In Figure 3.6, present computations for average skin-friction using the standard Wilcox k-ω, SST k-ω, SA and WA model are shown, the trends in results is similar to that in Figure 3.2; however, these results could not be compared with the experimental data since they are not given. It is not clear if they would show the same type of curve.

Figure 3.1 Time averaged pressure coefficient from leading edge to trailing edge of the circular cylinder at Re = 3.6 \times 10^6.
Figure 3.2 Time averaged skin friction coefficient from leading edge to trailing edge of cylinder at $Re = 3.6 \times 10^6$.
Figure 3.3 Time averaged pressure coefficient from leading edge to trailing edge of the circular cylinder at Re = $1 \times 10^6$. 

**Figure 3.3** Time averaged pressure coefficient from leading edge to trailing edge of the circular cylinder at Re = $1 \times 10^6$. 

- **Warschauer & Leene (ReD = 1.26 \times 10^6)** [16]
- URANS k-w standard
- URANS k-w SST
- URANS Catalano et al. [5]
- The WA model
Figure 3.4 Time averaged skin friction coefficient from leading edge to trailing edge of cylinder at $Re = 1 \times 10^6$. 
Figure 3.5 Time averaged pressure coefficient from leading edge to trailing edge of cylinder at $Re = 6.7 \times 10^5$. 
Figure 3.6 Time averaged skin friction coefficient from leading edge to trailing edge of cylinder at $Re = 6.7 \times 10^5$.

Table 3.1 provides a summary of the computed drag coefficient using various turbulence models and experimental drag coefficient results for the two Reynolds numbers $Re = 3.6 \times 10^6$ and $Re = 6.7 \times 10^5$ considered in this paper. Because of the oscillating nature of the wake flow, the average value of predicted drag coefficient as well as the experimental drag coefficient varies within a small range. From Table 3.1, WA model provides more accurate prediction compared to the SA model. Both WA model and SST $k$-$\omega$ model give satisfactory prediction of drag coefficient at $Re = 3.6 \times 10^6$. For smooth cylinder, at moderate Reynolds numbers from 1 to $10^3$, the flow begins to separate and starts behaving in a periodic manner by shedding asymmetric Karman vortices. As Reynolds number increases, the flow becomes fully separated and the boundary layer transition from laminar to turbulent takes place. As a result, there is a sharp drop in drag coefficient,
called the drag crisis at Reynolds number around $Re = 2 \times 10^5$. As Reynolds number further increases from $Re = 2 \times 10^5$ to $Re = 3.6 \times 10^6$, drag coefficient increases from around 0.4 to 0.7 because of the reattachment of the turbulent flow.

Table 3.1 Computed and measured $Cd$ for flow past a circular cylinder at various Re (“~” denotes unavailable data).

<table>
<thead>
<tr>
<th>Turbulence models (2D)</th>
<th>$Re(\times 10^6)$</th>
<th>$Cd$</th>
<th>$V$ (m/s)</th>
<th>$\phi$ (°)</th>
</tr>
</thead>
<tbody>
<tr>
<td>URANS SST k-ω</td>
<td>3.60</td>
<td>0.6500</td>
<td>22.04</td>
<td>108</td>
</tr>
<tr>
<td>URANS Standard k-ω</td>
<td>3.60</td>
<td>0.8153</td>
<td>22.04</td>
<td>117</td>
</tr>
<tr>
<td>URANS SA</td>
<td>3.60</td>
<td>0.3775</td>
<td>22.04</td>
<td>115</td>
</tr>
<tr>
<td>URANS WA</td>
<td>3.60</td>
<td>0.7470</td>
<td>22.04</td>
<td>119</td>
</tr>
<tr>
<td>URANS SST k-ω</td>
<td>1.00</td>
<td>0.7232</td>
<td>32.82</td>
<td>114</td>
</tr>
<tr>
<td>URANS Standard k-ω</td>
<td>1.00</td>
<td>0.4887</td>
<td>32.82</td>
<td>112</td>
</tr>
<tr>
<td>URANS SA</td>
<td>1.00</td>
<td>0.4186</td>
<td>32.82</td>
<td>108</td>
</tr>
<tr>
<td>URANS WA</td>
<td>1.00</td>
<td>0.8996</td>
<td>32.82</td>
<td>111</td>
</tr>
<tr>
<td>URANS SST k-ω</td>
<td>0.67</td>
<td>0.6867</td>
<td>32.82</td>
<td>112</td>
</tr>
<tr>
<td>URANS Standard k-ω</td>
<td>0.67</td>
<td>0.5321</td>
<td>32.82</td>
<td>113</td>
</tr>
<tr>
<td>URANS SA</td>
<td>0.67</td>
<td>0.4123</td>
<td>32.82</td>
<td>106</td>
</tr>
<tr>
<td>URANS WA</td>
<td>0.67</td>
<td>0.8996</td>
<td>32.82</td>
<td>115</td>
</tr>
<tr>
<td>Experimental data</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Roshko [6]</td>
<td>3.60</td>
<td>0.7</td>
<td>~</td>
<td>~</td>
</tr>
<tr>
<td>Roshko [6]</td>
<td>1.00</td>
<td>0.4</td>
<td>~</td>
<td>~</td>
</tr>
<tr>
<td>Roshko [6]</td>
<td>0.67</td>
<td>0.38</td>
<td>~</td>
<td>~</td>
</tr>
<tr>
<td>Achenbach [8]</td>
<td>3.60</td>
<td>0.76</td>
<td>~</td>
<td>115</td>
</tr>
<tr>
<td>Hoerner [10]</td>
<td>1.50</td>
<td>0.5</td>
<td>~</td>
<td>~</td>
</tr>
<tr>
<td>Hoerner [10]</td>
<td>2.00</td>
<td>0.6</td>
<td>~</td>
<td>~</td>
</tr>
<tr>
<td>Hoerner [10]</td>
<td>4.00</td>
<td>0.7</td>
<td>~</td>
<td>~</td>
</tr>
</tbody>
</table>

3.1.2 Contours from Numerical Simulations

In Figure 3.7 and Figure 3.8, computed time-averaged velocity and pressure contours respectively are presented at $Re = 3.6 \times 10^6$, $1 \times 10^6$ and $6.7 \times 10^5$ using the standard Wilcox k-ω, SST k-ω, SA and WA model. Qualitatively they exhibit the same pattern. Figure 3.9 shows the streamlines at a given instant of time during the transient simulations. There are differences
in the vortical flow patterns due to the fact that it is not possible to match an instant of time in various simulations.
Figure 3.7 Time averaged velocity contours averaged over 20 cycles.
Figure 3.8 Time averaged pressure contours averaged over 20 cycles.
(c) Standard $k$-$\omega$ at $Re = 3.6 \times 10^6$

(d) WA at $Re = 3.6 \times 10^6$

(e) SST $k$-$\omega$ at $Re = 10^6$

(f) SA at $Re = 10^6$

(g) Standard $k$-$\omega$ at $Re = 10^6$

(h) WA at $Re = 10^6$

(i) SST $k$-$\omega$ at $Re = 6.7 \times 10^5$

(j) SA at $Re = 6.7 \times 10^5$
3.2 Flow past a Square cylinder

3.2.1 Pressure Coefficients from Numerical Simulations

Computations are performed using the standard k-ω, SST k-ω and WA model and are compared with the experimental data reported by Bearman et al. [11], Lee et al. [12] and Nishimura et al. [13]. They are also compared with the DES simulations [14] and LES simulations [15]. Similar to the circular cylinder case, the criterion of convergence is that the drag coefficient oscillates periodically for more than 50 cycles. The results are time-averaged over more than 20 periods. The experimental results for pressure coefficient from leading edge to trailing edge show small difference from leading edge to the first corner of the cylinder. On the upper surface of the square cylinder, the pressure coefficient varies from -1.8 to -1.5 from different experiments. On the rear surface, the difference in pressure coefficient from various experiments is small as shown in Figure 3.10. Overall there is good agreement among the computations from various models and the experiments. Among the models, WA model gives the best match with the experimental data. Table 3.1 provides a summary of computed drag coefficient using various turbulence models and experimental drag coefficient result from Lyn et al. [2]. Because of the oscillating nature of the wake flow, the average value of predicted drag coefficient as well as the experimental drag coefficient varies within a small range. From Table 2, the experimental value of the drag...
The drag coefficient is in the range 1.9–2.2; all the models predict the drag coefficient in this range. The results for skin-friction could not be compared because it was not measured in the experiments.

![Figure 3.10](image)

**Figure 3.10** Time averaged pressure coefficient from leading edge to trailing edge of square cylinder at \( \text{Re} = 22000 \).

<table>
<thead>
<tr>
<th>Turbulence models (2D)</th>
<th>( \text{Re} \times 10^4 )</th>
<th>( \text{Cd} )</th>
<th>( U_\infty ) (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>URANS SST k-( \omega )</td>
<td>2.20</td>
<td>2.17</td>
<td>3.306</td>
</tr>
<tr>
<td>URANS Standard k-( \omega )</td>
<td>2.20</td>
<td>~</td>
<td>3.306</td>
</tr>
<tr>
<td>URANS SA</td>
<td>2.20</td>
<td>2.00</td>
<td>3.306</td>
</tr>
<tr>
<td>URANS WA</td>
<td>2.20</td>
<td>2.04</td>
<td>3.306</td>
</tr>
<tr>
<td>LES by Yong Cao [15]</td>
<td>2.20</td>
<td>2.02~2.77</td>
<td>~</td>
</tr>
<tr>
<td>LES by Sohankar et al. [19]</td>
<td>2.20</td>
<td>2.03~2.32</td>
<td>~</td>
</tr>
</tbody>
</table>
### 3.2.2 Contours from Numerical Simulations

In Figure 3.11 and Figure 3.12, computed time-averaged velocity and pressure contours respectively are presented at Re = 22,000 using the standard k-ω, SST k-ω, SA and WA model. Qualitatively they exhibit the same pattern. Figure 3.13 shows the streamlines at a given instant of time during the transient simulations. There are differences in the vortical flow patterns; this is probably because it is not possible to match an instant of time in various simulations.

![Velocity Contours](image)

Figure 3.11 Time averaged velocity contours (averaged over 20 cycles) at Re = 22000.
Figure 3.12 Time averaged pressure contours (averaged over 20 cycles) at Re = 22000.
Figure 3.13 Streamlines at one instant of time at Re = 22000.

3.3 Flow past a Curved Plate

3.3.1 Drag and Lift Coefficients from Experiment

For flow past a curved plate, both the drag crisis and the lift crisis were observed in a particular velocity range by Bot et al. [20]. For the circular cylinder, drag crisis occurs at Reynolds number $Re = 2 \times 10^5$, which is also called critical Reynolds number and the corresponding velocity is called the critical velocity. When the Reynolds number becomes higher, the separation point moves to a higher angle from the stagnation point of the cylinder. In the experiment [20] for flow around curved plate, the critical Reynolds number $Re_c = (2.00 \pm 0.04) \times 10^5$. A lift crisis is observed with lift-to-drag ratio jumping from -3 to +8.5 as shown in Figure 3.14. The time averaged lift coefficient jumps from -0.4 to +0.7 in a small Reynolds number range.
3.3.2 Lift and Drag Coefficients from Numerical Simulations

In a latest experiment by Bot et al. [20] for flow past nonsymmetrical objects, a lift crisis was observed in the drag crisis range of Reynolds numbers. Numerical simulations were performed using different models in FLUENT namely the laminar, the four-equation transition SST model, and the three-equation k-kl-ω model. Fully turbulent models like k-ω were applied at high Reynolds number but did not provide reasonable results. Velocity was adjusted to attain six different Reynolds numbers in range of $10^5 \sim 6 \times 10^5$. Like the previous two cases, the criterion of convergence was that the drag coefficient oscillates periodically for more than 50 cycles. The lift and drag coefficient were calculated by averaging over 20 periods. Averaged lift coefficient and drag coefficient from numerical simulation are shown in Figures 3.15 and 3.16. For the lift coefficient, transition models provide fairly good predictions in the entire Reynolds number range.

Figure 3.14 Lift coefficient $C_l$ (scale on the left axis) versus Reynolds number and drag coefficient $C_d$ (scale on the right axis) versus Reynolds number from the experiment [20].
As shown in Figure 3.15, especially in the lift crisis range, transition SST and k-kl-ω models provide lift coefficients which fit the experimental data reasonably well. When Reynolds number increases from $10^5$, the fluid flowing past the curved plate changes from laminar, transitional to turbulent, and the separation point moves downstream on the surface of the curved plate. The turbulent flow is much more robust than the laminar flow, it stays attached to the arc surface even at large pressure gradients. Laminar model was applied for Reynolds number from $1.5 \times 10^5$ to $2.5 \times 10^5$. It can be noticed that at Reynolds number $2.5 \times 10^5$, the lift coefficient computed by the laminar model has an error of 62.27%. When the Reynolds number is larger than the lift crisis range, the fluid undergoes transition from laminar to turbulent; the flow is no longer laminar, and this is the reason that numerical simulation using the laminar model cannot provide reasonable results. For the drag coefficient, the results in the transition region are not satisfactory. Transition SST model provides drag coefficient with reasonable trend, but the drag coefficient drops substantially at Reynolds number around $2 \times 10^5$. The numerical error for drag coefficient is distinctly higher than that for the lift coefficient.
Figure 3.15 Computed time averaged lift coefficient versus Reynolds number compared to the experiment lift coefficient versus Reynolds number.
Figure 3.16 Computed time averaged drag coefficient versus Reynolds number compared to the experimental drag coefficient versus Reynolds number.

To better understand the lift crisis, velocity distributions are shown in Figure 3.17, Figure 3.18, Figure 3.19, Figure 3.20, Figure 3.21 and Figure 3.22. Velocity profile at two positions $x/c = 1.2$ and $x/c = 1.5$ in Figure 3.17 show the streamwise velocity in the wake. On the upper surface of the plate, there is a separation point located around $x/c \approx 0.51$ [Figure 3.17(a) and Figure 3.18(a)]. As the Reynolds number increases, fluid behavior changes from laminar to transitional, and the drag crisis appears. The transition moves the separation point location downstream towards the trailing edge of the curved plate. The sharp change in separation point location is around Reynolds number
200,000 in the experiment of Bot et al [20]. The abrupt change in lift coefficient from negative to positive for Reynolds number around 200,000 is observed. From the time-averaged velocity contours, movement in separation point can be noticed from Reynolds number 180,000 to 220,000, which are close to the experimental transition Reynolds number. Flow remains attached to the arc surface for longer distance and the wake becomes narrower.

Figure 3.17 Velocity contours from different models and corresponding velocity profile at Reynolds number 150,000.
Figure 3.18 Velocity contours from different models and corresponding velocity profile at Reynolds number 180,000.
Figure 3.19 Velocity contours from different models and corresponding velocity profile at Reynolds number 220,000.

(a) k-kl-ω model

(b) Velocity profiles

(c) Transition SST model

(d) Velocity profiles
Figure 3.20 Velocity contours from different models and corresponding velocity profile at Reynolds number 250,000.

Figure 3.21 Velocity contours from different models and corresponding velocity profile at Reynolds number 400,000.
Figure 3.22 Velocity contours from different models and corresponding velocity profile at Reynolds number 600,000.

From the experiment [20], velocity profiles in the wake can be obtained. As the Reynolds number increases at around $Re = 2 \times 10^5$, the separation point moves towards the trailing edge and the
wake becomes narrower. The lowest normalized velocity at $x/c = 1.2$ increases from 0 to 0.5. Results at two positions of $x/c = 1.2$ and $x/c = 1.5$ are compared with experimental data. According to Figure 3.23 and Figure 3.24, the simulation results of velocity profile have good agreement with experimental data. The simulation results at $Re = 250,000$ have good agreement with experimental data at $Re = 205,000$. The results from simulation have small delay for the velocity prediction in the wake.
Figure 3.23 Velocity profile at $x/c = 1.2$ at different Reynolds number compared with experimental data.

Figure 3.24 Velocity profiles at $x/c = 1.5$ for different Reynolds number compared to the experimental data.
Chapter 4: Conclusions

In unsteady flow simulations of flow past a circular cylinder, a square cylinder and a curved plate using the URANS SA model, standard k-\omega model, SST k-\omega turbulence model and Wray-Agarwal (WA) model, WA model showed better accuracy compared to the SA model and was found to be competitive with the SST k-\omega model. For circular cylinder, separation point, pressure coefficient, skin friction coefficient and velocity contours were computed to make comparisons among various turbulence models and experimental data. All turbulent models provided satisfactory results for the pressure coefficient but big disagreement for the skin friction coefficient. For the square cylinder case, WA model provided more accurate prediction for the pressure on the cylinder surface compared to other turbulence models. WA model captured flow properties more accurately than the SA model.

For flow past an arc cylinder, a lift crisis found in the experiment [20] at Reynolds number of $2 \times 10^5$ was accurately simulated by k-kl-\omega and transition SST models. Near the critical Reynolds number, when the flow was undergoing laminar to turbulent transition, the laminar model, the SST transition model, the k-kl-\omega model and the k-\omega SST model were applied to simulate the transition. Both the SST transition model and the k-kl-\omega model predicted the lift crisis in reasonable agreement with the experimental data for the first time in the literature. However, k-kl-\omega model had slightly better overall predictions in the transitional regime. This problem requires further study to fully understand the flow physics behind the lift crisis. Further refinement of transition models as well as the LES model should be considered for this simulation.
References


Appendix

UDF of the WA model
//Wray-Agarwal Turbulence Model

#include "udf.h"
#include "mem.h"
#include "math.h"

#define Kappa 0.41
#define C1kw 0.0833 //kwConstant
#define Sigmakw 0.72 //kwdiffusion
#define C2kw (C1kw/Kappa/Kappa+Sigmakw) //kwConstant
#define C1keps 0.1127 //ProdConstant
#define Sigmakeps 1.0 //kepsdiffusion
#define C2keps (C1keps/Kappa/Kappa+Sigmakeps) //1.86 //kepsConstant
#define Cv1 13.0
#define MY_SMALL 1e-8
#define C_UDS(I,RG(c,t,i))C_STORAGE_R_NV(c,t,SV_UDS_I(i)+SV_UDS_0_RG-SV_UDS_0)

eenum{
    NuTilda,
    SRM,
    N_REQUIRED_UDS,
    D1,
    D2,
    Ebb,
    Eke,
    Ekw,
    Ekl,
    f1Switch_org,
    f1Switch_new,
    test1,
    test2,
    test3,
    test4
};

DEFINE_ON_DEMAND(setnames)
{
    Set_User_Scalar_Name(NuTilda,"NuTilda");
    Set_User_Scalar_Name(SRM,"SRM");
    Set_User_Memory_Name(D1,"D1");
Set_User_Memory_Name(D2,"D2");
Set_User_Memory_Name(Ebb,"Ebb");
Set_User_Memory_Name(Eke,"Eke");
Set_User_Memory_Name(Ekw,"Ekw");
Set_User_Memory_Name(Ekl,"Ekl");
Set_User_Memory_Name(f1Switch_org,"f1Switch_org");
Set_User_Memory_Name(f1Switch_new,"f1Switch_new");

DEFINE_ON_DEMAND(initialize)
{
    //TODO add check that the data exists to avoid crash
    Domain *d;
    Thread *t;
    cell_t c;

    d = Get_Domain(1);

    //thread loop
    thread_loop_c(t,d)
    {
        //cell loop
        begin_c_loop(c,t)
        {
            C_UDSI(c,t,NuTilda) = C_MU_T(c,t)/C_R(c,t);
            C_UDSI(c,t,SRM) = C_STRAIN_RATE_MAG(c,t);
        } //end cell loop
        end_c_loop(c,t)
    } //end thread loop
}

////////////////////////////////////////////////////////
///// FUNCTIONS /////////////
////////////////////////////////////////////////////////
DEFINE_ADJUST(adjust, d)
{
    Thread *t;
    cell_t c;
    real dist;
    real nu;
    real chi;
    real fv1;
    real eta;

    if (! Data_Valid_P())
    {

Message("\nNO DATA!\n")
return;
}

thread_loop_c(t,d)
{
begin_c_loop(c,t)
{

//Bound NuTilda and SRM to avoid divide by zero
C_UDSI(c,t,NuTilda) = MAX(C_UDSI(c,t,NuTilda),MYSMALL);
C_UDSI(c,t,SRM) = MAX(C_STRAIN_RATE_MAG(c,t),MYSMALL);

//Compute the switch function.
//TODO why does tanh fail when moved to an outside function call?
dist = C_WALL_DIST(c,t);
nu = C_MU_L(c,t)/C_R(c,t);
chi = C_UDSI(c,t,NuTilda)/(C_MU_L(c,t)/C_R(c,t));
fv1 = pow(chi,3.0)/(pow(chi,3.0)+pow(15.0,3.0));
//Original model f1 switch
C_UDMI(c,t,f1Switch_org) =
tanh(pow(MIN((C_UDSI(c,t,NuTilda)+nu)/(C_UDSI(c,t,SRM)*SQR(Kappa*dist)),SQR(C_UDSI(c,t,NuTilda)+nu)/SQR(C_WALL_DIST(c,t)))/0.4,4.0));

//New f1 switch function
//C_UDMI(c,t,f1Switch_new) =
tanh(pow(MIN(1.66*C_UDSI(c,t,NuTilda)/(Kappa*Kappa*C_UDSI(c,t,SRM)*SQR(dist)),SQR ((C_UDSI(c,t,NuTilda)+nu)/nu)),4.0));

eta = dist*sqrt(C_UDSI(c,t,NuTilda)*C_UDSI(c,t,SRM))/(20.0*nu);
C_UDMI(c,t,f1Switch_new) =
MIN(tanh(pow((1.0+20.0*eta)/(1.0+SQR(dist*MAX(sqrt(C_UDSI(c,t,NuTilda)*C_UDSI(c,t,SRM)),1.5)/(20.0*nu))),4.0)),0.9);

//Extra switches to test
C_UDMI(c,t,test1) = fv1;
//C_UDMI(c,t,test2) =
C_UDMI(c,t,test1)+0.3*(pow(C_UDMI(c,t,test1),6.0)-C_UDMI(c,t,test2)) ;
C_UDMI(c,t,test2) = 1.0-chi/(1.0+chi*fv1);
//C_UDMI(c,t,test3) =
C_UDMI(c,t,test2)*pow((1.0+pow(2.0,6.0))/(pow(C_UDMI(c,t,test2),6.0)+pow(2.0,6.0)),1.0/6.0);
C_UDMI(c,t,test3) = C_UDSI(c,t,NuTilda)/SQR(Kappa*dist)*C_UDMI(c,t,test2); 
C_UDMI(c,t,test4) = MIN(C_UDSI(c,t,NuTilda)/(C_UDMI(c,t,test3)*SQR(Kappa*dist)),10.0);
}
end_c_loop(c,t)
//Compute the reconstruction gradients
Alloc_Storage_Vars(d, SV_UDSI_RG(NuTilda), SV_UDSI_G(NuTilda), SV_NULL);
Scalar_Reconstruction(d, SV_UDS_I(NuTilda), -1, SV_UDSI_RG(NuTilda), NULL);
Scalar_Derivatives(d, SV_UDS_I(NuTilda), -1, SV_UDSI_G(NuTilda),
SV_UDSI_RG(NuTilda), NULL);
Alloc_Storage_Vars(d, SV_UDSI_RG(SRM), SV_UDSI_G(SRM), SV_NULL);
Scalar_Reconstruction(d, SV_UDS_I(SRM), -1, SV_UDSI_RG(SRM), NULL);
Scalar_Derivatives(d, SV_UDS_I(SRM), -1, SV_UDSI_G(SRM), SV_UDSI_RG(SRM),
NULL);

//Compute destruction terms based on reconstruction gradients
begin_c_loop(c,t)
    begin_c_loop(c,t)
        C_UDMI(c,t,Ebb) = MAX(NV_MAG2(C_UDSI_RG(c,t,NuTilda)), MYSMALL);
        C_UDMI(c,t,Eke) = MAX(SQR(C_UDSI(c,t,NuTilda)/C_UDSI(c,t,SRM))*NV_MAG2(C_UDSI_RG(c,t,SRM)), MYSMALL);
        C_UDMI(c,t,Ekw) = C_UDSI(c,t,NuTilda)/C_UDSI(c,t,SRM)*NV_DOT(C_UDSI_RG(c,t,NuTilda), C_UDSI_RG(c,t,SRM));
        C_UDMI(c,t,Ekl) = -1.0*C_UDMI(c,t,Ekw);
    end_c_loop(c,t)
end_c_loop(c,t)

//Free memory
Free_Storage_Vars(d, SV_UDSI_RG(NuTilda), SV_UDSI_G(NuTilda), SV_NULL);
Free_Storage_Vars(d, SV_UDSI_RG(SRM), SV_UDSI_G(SRM), SV_NULL);

real chi(cell_t c, Thread *t)
{
    return C_UDSI(c,t,NuTilda)/(C_MU_L(c,t)/C_R(c,t));
}
real fv1_15(cell_t c, Thread *t)
{
    real Chi = chi(c,t);
    return pow(Chi,3.0)/(pow(Chi,3.0)+pow(Cv1,3.0));
}

DEFINE_TURBULENT_VISCOSITY(mut_15,c,t)
{
    return C_R(c,t)*fv1_15(c,t)*C_UDSI(c,t,NuTilda);
}

DEFINE_SOURCE(source_prod,c,t,dS,eqn)
{
    dS[eqn] = (C_UDMI(c,t,f1Switch_new)*(C1kw-C1keps)+C1keps)*C_UDSI(c,t,SRM);
    return (C_UDMI(c,t,f1Switch_new)*(C1kw-C1keps)+C1keps)*C_UDSI(c,t,NuTilda)*C_UDSI(c,t,SRM);
}

DEFINE_SOURCE(source_dest,c,t,dS,eqn)
{
    return C_UDMI(c,t,f1Switch_new)*C2kw*C_UDMI(c,t,Ekw)-(1-C_UDMI(c,t,f1Switch_new))*C2keps*3.0*C_UDMI(c,t,Ebb)*tanh(C_UDMI(c,t,Eke)/(3.0*C_UDMI(c,t,Ebb)));
}

DEFINE_DIFFUSIVITY(diff_WAblend,c,t,eqn)
{
    real SigmaR = (C_UDMI(c,t,f1Switch_new)*(Sigmakw-Sigmakeps)+Sigmakeps);
    return C_MU_L(c,t)/C_R(c,t)+C_UDSI(c,t,NuTilda)*SigmaR;
}

DEFINE_UDS_FLUX(flux,f,t,i)
{
    real rho;
    
    if(BOUNDARY_FACE_THREAD_P(t))
    {
        if(NNULLP(THREAD_STORAGE(t,SV_DENSITY)))
        {
            //Boundary where density exists
            return F_FLUX(f,t)/F_R(f,t);
        }
    else
    {

//Boundary where density does NOT exist
rho = C_R(F_C0(f,t),THREAD_T0(t));
return F_FLUX(f,t)/rho;
}
}
else
{
  //Inner Face
  rho = 0.5*(C_R(F_C0(f,t),THREAD_T0(t)) + C_R(F_C1(f,t),THREAD_T1(t)));
  return F_FLUX(f,t)/rho;
}

////////////////////////////////////////////////////
// Boundary Conditions
////////////////////////////////////////////////////
DEFINE_PROFILE(inlet, t, i)
{
  face_t f;
  cell_t c0;
  Thread *t0 = t->t0;

  begin_f_loop(f,t)
  {
    c0 = F_C0(f,t);
    F_PROFILE(f,t,i) = 3*C_MU_L(c0,t0)/C_R(c0,t0);
  }
  end_f_loop(f,t)
}

DEFINE_PROFILE(outlet, t, i)
{
  //TODO add check for reversed flow, better definition of derivative.
  face_t f;
  cell_t c0;
  Thread *t0 = t->t0;
  int revFlowFaces = 0;

  begin_f_loop(f,t)
  {
    if(F_FLUX(f,t) < 0)
    {
      revFlowFaces = revFlowFaces++;
      c0 = F_C0(f,t);
      F_PROFILE(f,t,i) = 3*C_MU_L(c0,t0)/C_R(c0,t0);
    }
    else
    {
    }
  }
c0 = F_C0(f,t);
F_PROFILE(f,t,i) = C_UDSI(c0,t0,NuTilda);//looks like dNuTilda/dn=0
for orthogonal meshes
}
}
end_f_loop(f,t)

if(revFlowFaces > 0)
{
    //Message("nReversed flow on %i faces",revFlowFaces);
}
}
Vita

Ling Zhang

Degrees

M.S. Mechanical Engineering, May 2017
B.S. Mechanical Engineering, June 2014

August 2017