BBAA VI International Colloquium on:
Bluff Bodies Aerodynamics & Applications
Milano, Italy, July, 20–24 2008

INVESTIGATION INTO NUMERICAL MODELLING OF THE DRAG CRISIS FOR CIRCULAR CYLINDERS

Neil Botterill, Hervé Morvan, and John Owen

University of Nottingham
Department of Civil Engineering, Coates Building, University Park, Nottingham NG7 2RD, United Kingdom
e-mails: neil.botterill@nottingham.ac.uk, herve.morvan@nottingham.ac.uk, john.owen@nottingham.ac.uk

Keywords: Drag Crisis, CFD, LES

1 INTRODUCTION

Circular sections are used in a wide range of engineering applications in which they are subject to flow induced forces, from wind loads on lighting columns and chimneys to tidal and wave loads on offshore risers. As well as the classical vortex shedding phenomenon, circular sections are prone to several aero-elastic instabilities as seen in the galloping of bridge cables and the drag instability of tall lighting columns. These instabilities often occur at Reynolds numbers close to the drag crisis and the changes in flow regime associated with this transition may play an important part in driving the observed large amplitude oscillations.

The recent rapid advances in Computational Fluid Dynamics (CFD) have enabled increasingly complex fluid flow phenomena to be simulated. These simulations can now be coupled to models of structural response to predict fluid structure interactions and model various aero-elastic instabilities. The authors are currently applying these techniques to the flow around circular sections with the aim of simulating the instabilities described above. However, as noted, the aero-elastic instabilities of a circular section often occur at Reynolds numbers close to the drag crisis and it is therefore essential to accurately capture this event.

Given that the complexity of the flow at high Reynolds numbers makes direct numerical simulation prohibitively expensive, a key aspect in the simulation of flow around a bluff body is the appropriate choice of turbulence approach. Traditional Reynolds Averaged Navier-Stokes (RANS) models are unsuitable as unsteadiness forms an important feature of the flow in these problems. Although there are some notionally unsteady RANS (URANS) models available, these are not appropriate because it is still only the mean effects of turbulence, albeit in a transient framework, that are modelled, not the unsteadiness in the flow. Large Eddy Simulation (LES) offers an appropriate intermediate approach by allowing the direct simulation of the large scale unsteadiness, containing most of the turbulence energy, whilst retaining a closure model to represent the effects of smaller scale turbulence (Sub-Grid Scale model or SGS). However, the choice of this SGS model can have a critical effect on the accuracy of the simulations, especially close to boundaries. In this paper the authors consider the use of two different SGS models in the LES of the flow around a circular cylinder, and compare their effectiveness in predicting the drag crisis.

2 METHODOLOGY

The commercial CFD solver used for all the numerical simulations presented here is ANSYS CFX. The investigations will focus on the use of a LES formulation based on two types of SGS model: (1) the well established Smagorinsky [1] model, based on an algebraic turbulence viscosity dependent on a constant parameter C_S which tends to be problem specific and needs to be damped at the wall; (2) the Dynamic model (Germano et al. [2]; Lilly [3]) for which the parameter C_S is evaluated locally based on the comparison of the sub-grid tensor values computed using two different filters. A detailed description of each of these turbulence models will be given in the full length paper, however a key feature of the Dynamic approach relevant to the present work is that it does not require any artificial wall damping and is able to capture the transition process (Wagner et al. [4]). This is important as the prediction of the drag crisis in the cylinder case is dependent on the position of the separation point. The role and ability of the SGS model to predict what is happening at the walls is therefore another key consideration in the choice of the turbulence formulation. For each of the turbulence SGS models, simulations were performed for a range of Reynolds numbers within which the drag crisis is expected to occur. For each simulation, time histories of the drag force and lift force experienced by the cylinder were recorded. Mean and RMS values of lift and drag together with estimates of the Strouhal number were used to compare the two SGS models, and evaluate their performance against published experimental data. The domain dimensions in terms of cylinder diameter d are: $18d$ in the streamwise direction, $8d$ in the transverse direction, and $2.5d$ in the spanwise direction. The cylinder axis is positioned $4d$ away from the inlet.

3 RESULTS AND DISCUSSION

Figure 1 contains plots of drag coefficient produced by both the LES Smagorinsky and LES Dynamic turbulence models for Reynolds numbers in the range $40,000$ to $400,000$. It can be seen that the Smagorinsky model does not predict a drop in drag consistent with the drag crisis seen in the published data, Figure 2, whereas the Dynamic model predicts a drop in drag from 1.1 to 0.5 in a Reynolds number range of $60,000$ to $100,000$. The latter result is encouraging; however the drop predicted by the Dynamic model approach occurs for a lower Reynolds number range and the drag crisis line appears to be shifted to the left. Its slope is also less steep that the curve gradient visible in Figure 2 and the predicted drag coefficient does not drop as low. However, the ESDU data [6] illustrate that the drag crisis is sensitive to inlet turbulence and surface roughness around the cylinder, and there are experimental data that indicate that the drag crisis can occur at a lower Reynolds number due to blockage effects (Richter and Naudascher [7]). Other experimental literature discusses the effect of varying the spanwise extent of the model (Norberg [8]). Thus the importance of a correct choice of domain size is highlighted.

The reason for the improved performance of the dynamic model over the Smagorinsky SGS model can be seen by considering the flow fields in the near wake region, Figures 3a and 3b. Zdravkovich [5] describes the process of the drag crisis in terms of the point of transition to turbulence found in the cylinder wake. The transition to turbulence occurs closer to the rearward face of the cylinder as the Reynolds number increases. At a Reynolds number just below that

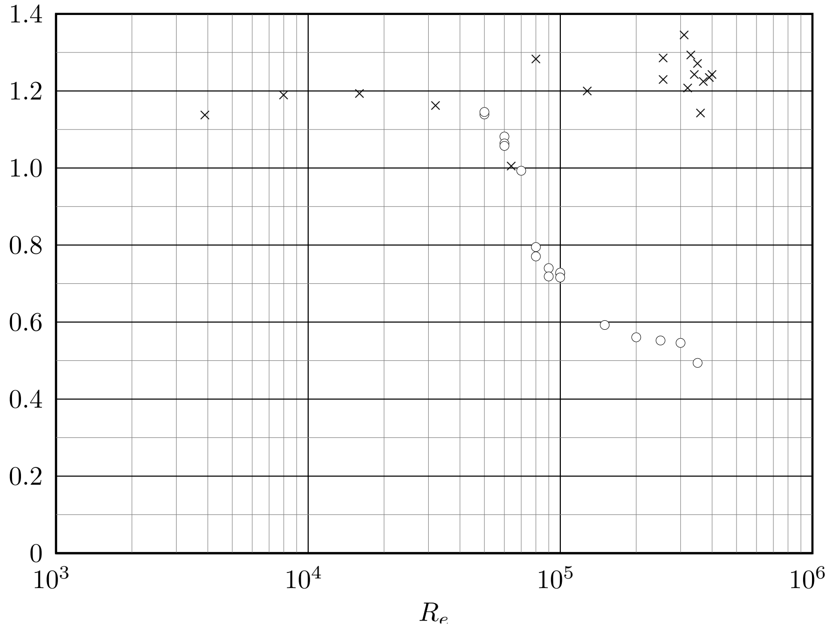


Figure 1: Average drag data from the numerical simulations
(× Smagorinsky; ○ Dynamic)

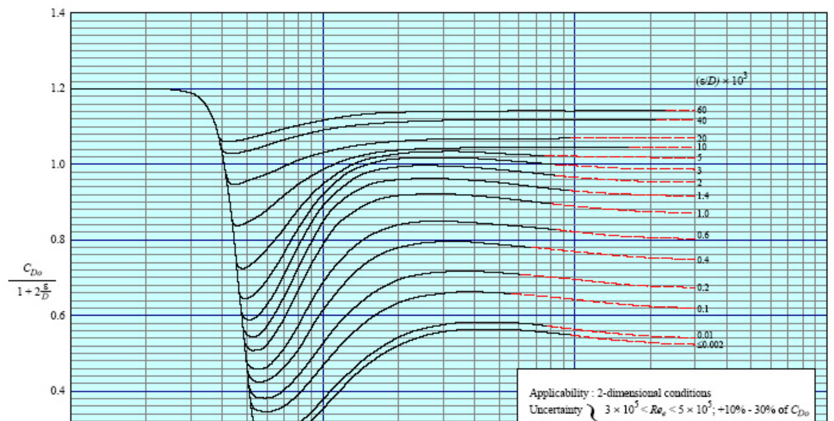
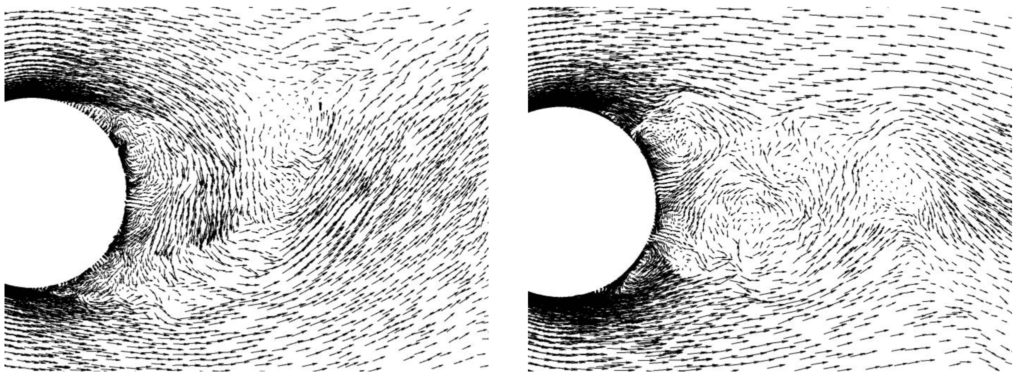


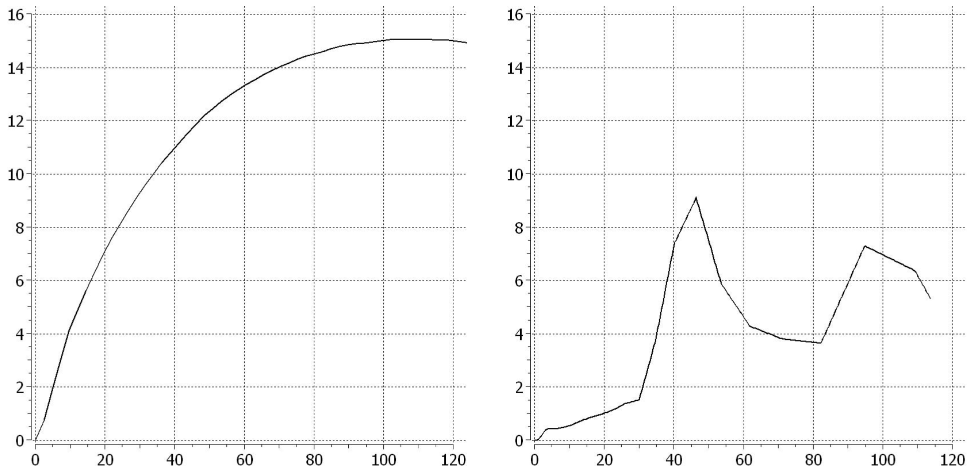
Figure 2: ESDU plot of mean drag through the drag crisis

for the drag crisis two transition points can be observed, one in each shear layer just beyond the point of separation on either side of the cylinder. An increase in Reynolds number causes these points to move causing one or both separation points to jump from a position at roughly $90°$ to a position $120°$ on the perimeter measured from the upstream stagnation point. This results in a drop in drag caused by the narrowing of the wake behind the cylinder. This happens, to an extent, with the Dynamic model, Figure 3b, but it does not materialise with the Smagorinsky model, Figure 3a. Plotting the turbulence viscosity in the first part of the boundary layer, Figure 4, reveals why this is the case. The Smagorinsky closure does not recover a sensible asymptotic behaviour of the sub-grid viscosity at the wall.



(a) Smagorinsky (b) Dynamic

Figure 3: Typical velocity flowfield vector plot at the rearward side of the cylinder



(a) Smagorinsky (b) Dynamic

Figure 4: Typical plots of eddy viscosity ratio against y^+ at cylinder wall normal

REFERENCES

[1] J. Smagorinsky: *General circulation experiments with the primitive equations*, **91**, 99–164.

[2] M. Germano, U. Piomelli, P. Moin, W.H. Cabot: *A dynamic subgrid-scale eddy viscosity model*, Physics of Fluids A: Fluid Dynamics, **3**, 1760–1765.

[3] D.K. Lilly: *A proposed modification of the Germano subgrid-scale closure method*, Physics of Fluids A: Fluid Dynamics, **4**, 633–635.

[4] C.A. Wagner, T. Hüttl, P. Sagaut: *Large Eddy Simulation for Acoustics*, Cambridge University Press, 2007

[5] M.M. Zdravkovich: *Flow Around Circular Cylinders Vol. 1*, Oxford Science Publications, 1997

[6] Engineering Sciences Data Unit: *ESDU 80025: Mean forces, pressures and flow field velocities for circular cylindrical structures*.

[7] A. Richter, E. Naudascher: *Fluctuating forces on a rigid circular cylinder in confined flow*, Journal of Fluid Mechanics, **78**, 561–576.

[8] C. Norberg: *Fluctuating lift on a circular cylinder: review and new measurements*, **17**, 57–96.