NUMERICAL DEVELOPMENTS IN UNSTEADY AERODYNAMIC FLOWS

D. Drikakis* and G. Barakos

*Queen Mary and Westfield College (QMW), Engineering Department, University of London, Mile End Road, London E1 4NS, United Kingdom
e-mail: d.drikakis@qmw.ac.uk web page: /http://www.eng.qmw.ac.uk

Key words: Unsteady Flows, Dynamic Stall, Oscillating/Pitching Aerofoils, Buffet, Flutter, Shock-diffraction, Riemann solvers, Implicit schemes, Turbulence Modelling

Abstract. The paper summarises recent developments pursued by the unsteady aerodynamics group at QMW. These include the development of computational fluid dynamics methods and validation of turbulence models for essentially unsteady flows with moving boundaries as well as investigation of various unsteady aerodynamic phenomena including dynamic stall, buffet, flutter and unsteady shock-wave diffraction. Overall, these studies have so far shown that simulations of unsteady aerodynamic flows which are strongly affected by turbulence, can provide a fair qualitative picture in most of the cases, but there are still many open issues regarding the quantitative agreement between experiments and simulations. Particular challenges and remaining open questions appear in the case of unsteady flows around pitching and oscillating aerofoils featuring dynamic stall. The various results indicate some strengths of the Godunov-type methods, implicit solvers, as well as linear, non-linear (eddy-viscosity-based) and Reynolds-stress models employed in this study, but they also reveal weaknesses associated in particular with modelling issues of high Reynolds number compressible flows.
1 INTRODUCTION

In the context of aerodynamics, the numerical simulation of unsteady and compressible flows around moving boundaries is motivated by the need to understand flow phenomena associated with the behaviour of aircrafts during manoeuvres, flows around helicopter rotors and turbomachinery blades, flows around wind generators, as well as flows with strong shock waves for which the applications are spanning from engineering to astrophysics. The flow phenomena associated with the above applications include separation, unsteadiness, shock/boundary-layer, viscous/inviscid, vortex/body, and vortex/vortex interactions, shock-wave reflection and diffraction as well as flow instabilities, transition to turbulence and flow re-laminarisation. A better understanding of the fundamental flow physics in the aforementioned applications will possibly enable us to control the flow and, subsequently, enhance the performance of aircrafts and helicopters.

The cost of performing wind tunnel or flight experiments of unsteady flows is very high. Moreover, the information obtained through experiments is usually limited - due to the instrumentation constraints - to pressure distributions or aerodynamic coefficients. Numerical simulation of unsteady flows is a promising alternative, since it can provide more detailed information than experiments. However, it is not free of shortcomings and difficulties because of numerical and turbulence modelling limitations. The present paper summarises research efforts of the QMW’s unsteady aerodynamics group in connection with the development and implementation of CFD methods in unsteady aerodynamic flows.

Concerning numerical methods, the challenges are associated with the accuracy and efficiency of the discretisation scheme and iterative solver in time accurate computations. Particular difficulties arise in the case where the numerical method is used in conjunction with low-Re number turbulence models whether based on the eddy-viscosity concept or second moment closures. State-of-the-art numerical methods used in the simulation of compressible flows have principally been developed for the Euler equations of gas dynamics. These methods include second- and third-order Godunov-type methods, and so far did not account for turbulence quantities, but only for the mean flow variables. Yet, in the case of unsteady flows it is of particular importance to strongly couple the turbulence and mean flow equations for reasons of both efficiency and accuracy. This motivated efforts of the present group to progress beyond the current state of the art and strongly couple the partial differential equations of the turbulence models with the Navier-Stokes equations through an implicit unfactored method and a high-resolution Riemann solver.

Significant efforts have been spent by the group in connection with validation and assessment of turbulence models in unsteady aerodynamic flows. The state of the art in the literature has so far been based on algebraic, one-equation and high-Re $k - \epsilon$ and $k - \omega$ models. Virtually all turbulence models have been formulated for and calibrated by reference to steady flows. Modelling unsteady turbulent flows presents particular challenges. These are concerned, specifically, with the application of closure assumptions with trans-
port associated with high rates of changes, and with capturing the lag between mean-flow and turbulence quantities, especially at high frequencies. This suggests the need for advanced modelling practices. Therefore, in the present project non-linear eddy-viscosity models (EVM) and second-moment closure were implemented. However, significant efforts were also spent in assessing several variants of low-Re linear eddy-viscosity models (EVMs).

Finally, due to the limited length of the manuscript, only a couple of indicative results from the group’s research on unsteady shock-wave phenomena is presented here, and for more information we refer the reader to the relevant publications cited in the end of the paper.

2 NUMERICAL AND PHYSICAL MODELLING

Unsteady aerodynamic phenomena are highly non-linear and, therefore, cannot be described using linear aerodynamic theory [1]. In view of this remark, an appropriate set of governing equations to be employed is the complete Reynolds-averaged, compressible Navier-Stokes equations. Additional equations are required to model turbulence transport. Furthermore, if the response of the aerodynamic structure is of interest, then linear structural analysis equations have to be coupled with the above [2].

The time averaged Navier-Stokes equations for a compressible fluid can be written as:

$$\begin{align*}
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) &= 0, \\
\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i u_j + p \delta_{ij} - \tau_{ij}) &= 0, \\
\frac{\partial p e}{\partial t} + \frac{\partial}{\partial x_i} (p u_i (e + p) - u_i \tau_{ij} - \dot{q}_i) &= 0.
\end{align*}$$

In the above equations, the ideal gas equation of state \( p = \rho RT \) is used to calculate the pressure, \( p \), while \( \tau_{ij} \) is obtained by the sum of the laminar and Reynolds stress tensors.

To close the above system of equations the definitions of the turbulent Reynolds stresses and heat fluxes (\( \dot{q}_i \)) as functions of the mean flow quantities is required. For linear eddy-viscosity models (EVM), the stress tensor is defined, according to the Boussinesq approximation i.e. to be proportional to the mean strain-rate tensor, with the factor of proportionality being the eddy viscosity (\( \mu_T \)). The eddy viscosity is modeled in terms of the turbulent kinetic energy and a turbulence scale variable, the latter being dependent on the model employed.

To model the structural response of a lifting surface a two degree of freedom system can be considered. For a rigid aerofoil, having two degrees of freedom in pitch (\( h \)) and plunge (\( \alpha \)) the governing equations are:

3
\[ M \frac{d^2 q}{dt^2} + K q = F \] (2)

\[ q = \begin{bmatrix} \frac{h}{b} \\ \alpha \end{bmatrix}, \quad M = \beta \begin{bmatrix} 1 & x_\alpha \\ x_\alpha & r_\alpha^2 \end{bmatrix} \] (3)

\[ K = \beta \begin{bmatrix} \omega_R^2 & 0 \\ 0 & r_\alpha^2 \end{bmatrix}, \quad F = \beta \begin{bmatrix} -C_l \\ \frac{-2C_m}{2C_m} \end{bmatrix}, \quad \beta = \frac{4}{\pi \mu \omega_\alpha^R} \] (4)

where: \( \alpha \) is the aerofoil’s incidence angle, \( h \) is the aerofoil’s displacement in the vertical axis, \( b = c/2 \) is the aerofoil’s half-chord, \( \omega_R = \omega_h/\omega_\alpha \) is the ratio of natural frequencies in plunge and pitch, \( x_\alpha \) is the offset of the center of gravity from elastic axis, \( r_\alpha \) is the radius of gyration of the torsional spring, \( C_l \) is the lift coefficient, \( C_m \) is the moment coefficients about the elastic axis, \( \mu = m/(\pi \rho_\infty \beta) \) is the ratio of aerofoil to fluid mass, \( \omega_\alpha^R \) is the natural frequency of pitching, dimensioned using \( U_\infty/2b \), and \( \overline{U} = 2/\omega_\alpha^R \) is the reduced velocity derived from \( \omega_\alpha^R \). The above equations are standard for the aeroelastic analysis of a pitching and plunging aerofoil [2]. They are in fact a system of ordinary differential equations that can be solved using any method for integration of ODEs, like a multi-step Runge-Kutta method, provided that the aerodynamic loads \( C_l \) and \( C_m \) are known.

A brief description of the numerical method employed for the general case of unsteady and turbulent flow around a moving boundary, is given here. A detailed description can be found in references [3, 4]. The present scheme solves the conservation equations of mass, momentum and energy along with the turbulence-transport equations using a finite volume approach and body-fitted curvilinear coordinates \((\xi, \zeta)\). A third-order upwind scheme is also used in conjunction with a Riemann solver in order to increase the accuracy of the inviscid fluxes calculation at the cell faces of the computational volumes [5, 6, 7]. Limiters based on the square of pressure derivatives have been used for detecting shocks and contact discontinuities. The viscous terms are discretized by central differences.

The time-accuracy can be obtained by using either implicit or explicit schemes, and the efficiency of the above is strongly related to the time scales imposed by the prescribed motion of the solid boundaries. Both explicit and implicit schemes have been implemented into the CFD code [8]. In the present study the implicit version of the method has been employed according to which the six or eight equations - in the case of RSTM - are simultaneously solved by an implicit-unfactored method which combines Newton sub-iterations and Gauss-Seidel relaxation. This approach solves all equations in a strongly coupled manner using an implicit-unfactored relaxation scheme. This results in a compact numerical implementation, while requires a moderate number of Newton iterations at each time step. However, the implementation is fairly complicated and also requires a realistic
D.Drikakis and G. Barakos

initial guess before the time-marching is initiated [8]. For the implicit solution of the governing equations, a second-order approximation of the time derivative has been used. For every cell of the computational domain, a diagonally dominant system of six to eight equations, depending on the closure employed, is obtained and subsequently inverted by a matrix-inversion technique. This system is solved here by a point Gauss-Seidel scheme. Moreover, to obtain high values of the CFL number, preconditioning is also performed at each Gauss-Seidel sub-iteration.

3 COMPUTATIONAL RESULTS

3.1 Numerical investigation of dynamic stall phenomena

The dynamic stall (DS) can be studied by considering the pitching motion of an aerofoil beyond its static-stall incidence angle. There are several phenomena associated with the pitching motion of an aerofoil, the most important being the generation of intense vorticity over the suction surface near the leading edge. As the pitching motion goes on, the vorticity generation results in a discrete vortex which detaches from the body and convects along the suction surface. Subsequently, this leads to large variations of lift, drag and pitching moment coefficients.

As can be seen in Figure 1, in the case of a harmonically oscillating aerofoil there is hysteresis of the aerodynamic loads during the motion of the lifting surface even for a small amplitude of oscillation. The results in this figure have been obtained by using selective models from three major model categories: linear EVM model, non-linear EVM (NLEVM) and Reynolds-stress transport model (RSTM). The NLEVM provides slightly better results than the linear $k-\epsilon$ model for the moment coefficient, while there are very small differences between the RSTM and NLEVM. The variation of the aerodynamic loads during the motion can be larger as the amplitude of oscillation increases. This can be seen in Figure 2 where the aerodynamic loads of a NACA-0012 oscillating in deep stall conditions are presented. The results are compared with experimental data of McAlister et al. [9]. The free stream Reynolds and Mach numbers are $10^6$ and 0.2, respectively. The oscillation amplitude is $10^\circ$ around a mean incidence angle of $15^\circ$, and the reduced frequency of the oscillation is $k_f = 0.25$. As can be seen from the predicted unsteady airloads, the linear model (LS) fails to predict essential features of the loop curves, e.g. the “double-loop” of the moment curve (Figure 2). This is due to the fact that the linear model leads to an excessive production of turbulence that inhibits flow separation, thus resulting in poor predictions of the lift and moment loops. In this case, a better prediction has been obtained using the non-linear eddy-viscosity model (also labelled NL).

During the ramping motion of a lifting surface a vortex emerging from the leading edge region of the aerofoil, grows in size as the incidence increases (Figures 3 and 4). This vortex is identified as the Dynamic Stall Vortex (DSV). At higher incidence angles there are two other vortices formed at the leading edge upstream of the DSV and another rapidly growing vortex emerges from the trailing edge. At maximum incidence, both the DSV
Figure 1: Comparisons between linear, non-linear eddy-viscosity models and second-moment closures for the AGARD CT-1 Case [10]; NACA0012 aerofoil at $Re = 4.8 \times 10^6$ and $M = 0.6$, with mean incidence angle $2.8^\circ$, amplitude of oscillation $2.4^\circ$ and reduced frequency of $0.16$. The linear EVM model by Fan et al.[11] (FLB), the NLEVM by Craft et al.[12] and the RSTM by Launder and Shima [13] have been employed for this case.

and the trailing edge vortices detach from the aerofoil. For the harmonically oscillating NACA-0012 aerofoil at high-Reynolds numbers and deep stall conditions (Figure 2), the onset of the DSV is not different from the ramping case. However, for the oscillating aerofoil the DSV is shed at the wake as the aerofoil begins the downstroke motion, and the flow starts to reattach at the leading edge. The vortical structures formed around the aerofoil at maximum incidence, both for the ramping and harmonically oscillating cases, are shown in Figure 5.

Figure 2: Comparison between linear and non-linear eddy-viscosity models for the unsteady airloads of the NACA-0012 aerofoil.
Figure 3: Density field around a ramping NACA-0012 aerofoil ($Re = 10^5, M = 0.25, \alpha^+ = 0.1, \alpha_{max} = 60^\circ$)
Perhaps the most important parameter in unsteady flows around pitching and oscillating aerofoils is the speed of the induced motion of the profile, since this constitutes the driving mechanism of the DS phenomenon. For high pitch rates, there is an almost linear relationship between pitch rate and stall angle. The same is true for low pitch rates, but the linear relationship has a different slope. However, there is a certain region of pitch rates where the two curves intersect, marking clearly that, at low pitch rates, there is a
definite change of the phenomena associated with the unsteady motion of the profile.
Figure 6 presents the lift curves as well as the instantaneous surface pressure distribution
at stall angles, and at three different pitch rates of $k_f = 1.5, 0.5$ and $0.1$, respectively.
Results have been obtained using the NLEVM by Craft et al. [12] (NLEVM) and the
Spalart-Allmaras [14] model (SA). The main finding of the calculations was that for pitch
rates less than 0.5 the DSV was not present during pitch. Instead, a trailing edge vortex
was present as it is also indicated by the the instantaneous $C_p$ curve at stall conditions
(Figure 6(b)). Moreover, at even lower pitch rates (Figure 6(c)) periodic shedding of
vortices from the trailing edge was encountered, subsequently leading to oscillating aero-
dynamic loads.

For all cases the stall angle was determined based on the change of sign of the derivative
of the lift and moment curves and these results are shown in Figure 7 using the NLEVM
and Spalart-Allmaras models. The computations reveal a change of the curve slope,
for lower values of the pitch rate, and this is in agreement with Wilby’s findings [15].
Furthermore, the time histories of the pressure-coefficient distributions suggest that there
are differences in the DSV onset between lower and higher pitch rates. In the former
case a vortex is formed at the trailing edge of the profile as soon as the aerofoil reaches
a certain incidence angle. The shedding of this vortex in the wake marks the stall of the
aerofoil. This is very similar to the static stall of an aerofoil. On the other hand, at higher
pitch rates the DSV formed near the leading edge of the profile propagates downstream
over the suction side of the airfoil and is finally shed in the wake.

### 3.2 Computation of transonic buffeting flows

Flow unsteadiness around a lifting surface may originate from the surface motion itself
or from unsteady boundary conditions. However, there are cases that certain flow condi-
tions, i.e. combination of Reynolds and Mach numbers, can also induce unsteadiness. In
the case of transonic buffet, the self-excited shock oscillations are followed by flow sepa-
ration. It is thus important for a turbulence model to predict accurately the separation
induced by the interaction of the shock with the boundary layer and, subsequently, the
buffet onset.

McDevitt and Levy [16] have performed several wind tunnel tests for the Shock Induced
Oscillation (SIO) of the 18 % circular arc aerofoil. In the relevant references the SIO was
reported to occur at Mach numbers between 0.73 and 0.78 and for Reynolds numbers
between 1 and 15 million. In the experiments considered here special care has been
taken to simulate free air streamline conditions by contouring the wind tunnel walls. The
characteristic frequency of the oscillations is reported to be 0.47 varying little with Mach
number.
McDevitt and Okuno [17] have also performed experiments for the NACA-0012 aerofoil at Mach numbers between 0.7 and 0.8, angles of incidence less than 5 degrees and $Re_c$ number between $10^6$ and $14 \times 10^6$. For the buffeting onset they identified the incidence angle and Mach number as the most important parameters. Their wind-tunnel results
Figure 7: Numerical predictions of the variation of the stall angle with the pitch rate for a ramping NACA-0012 airfoil at $M = 0.302$, $Re = 10^6$, $\alpha_0 = -4^\circ$, $\alpha_1 = 30^\circ$.

are especially suitable for validation of CFD codes since they are free of wall effects in contrast to previous experimental studies [16].

For the 18% circular-arc aerofoil computations have been performed using a two-block grid consisting of $120 \times 80$ cells per block. The blocks were placed symmetrically around the top and bottom surface of the aerofoil. Computations were performed for two Mach numbers 0.77 and 0.72, while the Reynolds number remained constant at $10^6$ and the incidence angle was zero degrees. Iso-density contours are used in Figure 8 to show the relative motion of the shocks formed on both the suction and pressure sides of the aerofoil. The startup of the SIO is shown in Figure 9, where the time history of the lift coefficient is presented. Results are presented here for two cases using the linear $k - \omega$ model [18]. The limit cycle develops after three oscillations, approximately, and the amplitude of the cycle remains constant thereafter (Figure 9(a)). For this case, the flow after the shock is separated. Underprediction of the separation will eventually result in steady flow. The separation, however, is not only due to the shock/boundary-layer interaction but to a great extent it is also due to the favourable curvature of the surface. Decreasing the Mach number was found to be sufficient to eliminate the limit cycle as can be seen from Figure 9(b). The calculated reduced frequency of the shock oscillations was found to be $k = 0.444$ while the experiments by McDevitt and Levy [16] reported a value close to 0.47. The shocks oscillate in the trailing region of the profile over an area about 40% of the aerofoil’s chord and this is in agreement with the experimental observations. The main conclusion from this calculation is that although different turbulence models may predict different frequencies and different shock positions, all of them were able to predict
some form of SIO instead of a steady flow.

Figure 8: Instantaneous density contours around the 18% circular-arc aerofoil.

Figure 9: Buffet onset - 18% circular arc aerofoil ($Re = 2 \times 10^6$, $\alpha = 0^\circ$).

In order to compute the transonic buffeting flow around a NACA-0012 aerofoil, several computational grids have been employed to ensure grid-independent solutions[19]. In addition, calculations have been undertaken for various dimensions of the computational domain to ensure independence of the solution from the far-field boundary conditions. In Figure 10, comparison of numerical and experimental results for the buffet onset is presented. There is a well-defined region of Mach and incidence angle where buffet occurs.
Initially, four computations (Figure 10) were performed at conditions below the experimentally reported buffet onset and steady-state solutions were achieved (symbols in Fig. 10 labeled “no SIO (shock-induced oscillation)”). Afterwards, the incidence-angle was slowly increased to obtain unsteadiness and it was found that the unsteady computations resulted in either periodic loads, thus indicating buffet (symbols in Fig. 10 labeled “SIO”), or in steady-state flow. In the latter case, the computations were repeated for a higher incidence-angle until buffet is captured. Once buffet was predicted, the incidence-angle was again decreased and the computation was repeated to check whether the experimental boundary (solid line in Figure 10) for buffet onset could be closer approached.

For all combinations of Mach number and incidence angle considered here, the linear $k - \epsilon$ models led to a steady solution, thus, failing to predict buffet. As can be seen in Figure 10, the computations predict the buffet onset boundary slightly shifted to higher incidence angles and Mach number. This is in agreement with the calculations of Leballeur and Girodroux [20]. Edwards [21], however, reported results closer to the experimental data using an inverse boundary-layer method and the Baldwin-Lomax (1978) model. He also found that buffet occurs at $\alpha = 0^\circ$ and Mach number close to 0.83.

![Figure 10: Buffet onset for the NACA 0012 aerofoil ($Re_c = 10^7$, $M = 0.775$, $\alpha = 4^\circ$). Solution obtained using the SA model [14] (crosses) and the non-linear $k - \omega$ model [22] (squares); Experimental data from McDevitt and Okuno [17].](image)

3.3 Flutter analysis of wing sections

Flutter is classified into various types depending on the aerodynamic conditions. It is known that as the incidence of an aerofoil exceeds the stall angle, then the lift rapidly decreases. The aerodynamic mechanism of stall is not explainable by linear aerodynamic theory since massive separation occurs. In many instances the flow separation occurs in an unstable way under the presence of flutter. If a body encounters flutter and during flutter the flow is separated, then the phenomenon is called stall flutter [23]. Stall flutter
is very important in rotating machinery like propellers, turbine blades and compressors which operate at incidence angles close to the static stall angle. However, as modern wing designs tend towards thinner wing sections and large wing spans, stall flutter is also added to the design concerns.

The decoupled approach has been utilised for the prediction of the stall-flutter of a NACA-0012 aerofoil. As suggested by Clarkson et al. [24], a NACA-0012 aerofoil harmonically oscillating about 9.97° with an amplitude of 9.88°, and flow conditions corresponding to $Re = 3.67 \times 10^6$ and Mach number 0.302, was employed as a test case to investigate stall flutter. McCroskey et al. [9] have performed experiments for this case and their experimental data indicate multiple loops on the $C_m$ curve. At first, the unsteady aerodynamic loads were predicted using the numerical technique described in [3]. Attempts to simulate this flow with the Spalart-Almaras [14] model has shown that multiple loops can be obtained only by tripping the flow. This was in agreement with the results obtained by Clarkson et al. [24] using the Baldwin-Lomax [25] model. The comparison between computations and experiments for the aerodynamic loads are given in Figure 11. The same figure shows that results as obtained by using the semi-empirical ONERA model [26].

Figure 11: Unsteady aerodynamic loads as predicted using the SA model for the oscillating NACA-0012 aerofoil ($M = 0.302$, $Re = 3.67 \times 10^6$, $\alpha_0 = 9.97^\circ$, $\alpha_1 = 9.88^\circ$). The predictions correspond to the fully-turbulent and transitional calculations; The fit of the aerodynamic loads as obtained using the ONERA model is also presented.

The simulation results for stall flutter cases were further utilised to estimate the flutter velocity and frequency for a wing having material parameters as specified in [27]. The flutter computation is based on three steps. At first the unsteady aerodynamic loads for an oscillating wing section are obtained. In a second stage, the semi-empirical ONERA
aerodynamic model developed by Tran and Petot [26] is used to fit the obtained aerodynamic loads. Finally, the method by Kim and Dugundji [27] is employed to obtain the flutter frequency for a wing section having specified material parameters. A detailed description of the procedure can be found in [28].

Figure 12 shows the variation of the oscillation amplitude with frequency and velocity, respectively, at flutter conditions at the tip of the fluttering wing. Results from the work of Kim and Dugundji [27] for a similar solid model predict flutter frequency of about 20 Hz. The present results predict a lower frequency than the analysis of Kim and Dugundji [27]. However, this was expected since a different approach regarding modelling of the aerodynamic loads was employed in the present study.

Under transonic flow conditions the interactions of the shock waves with the boundary layers, on both sides of the aerofoil, lead asymptotically to limit cycle behaviour after a bifurcation from the stable undisturbed state. In addition, different types of instability can interact and an example of this interaction could be the flutter-divergence interaction. The non-linear aeroelastic analysis of a NACA64A006 aerofoil has been considered as a test problem by many authors [29, 30]. The structural parameters suggested in the above two works are also employed here. For this case a strongly coupled solution of the fluid-flow and structural analysis equations is used [31]. The structural model is a standard two degree of freedom system.

Reasonable spatial accuracy has been obtained using a mesh of 180 × 90 cells. The far field boundary position was also found to be important for this problem and thus investigations have been undertaken using \( \overline{U} = 1.7, M_\infty = 0.92 \) and \( Re = 10^6 \). A smooth periodic response has been obtained for computational grids having the far-field boundary at about 15 chord-lengths far from the aerofoil’s surface. The response of the pitch angle
for various CFL numbers is shown in Figure 13(a). The CFL number of 100 resulted in about 70 steps per oscillation cycle for the implicit method. The amplitude of the oscillation for various $\overline{U}$ values is shown in Figure 13(b). The present results for the oscillation amplitude are in reasonable agreement with the results of Bendiksen and Kousen [29] obtained for a similar case.

3.4 Unsteady shock-wave phenomena

The group has been engaged over the past few years in a number of investigations related to unsteady shock-wave phenomena. These include investigation of unsteady shock wave diffraction around cylinders [7, 32, 33] including blast-wave modelling effects [34]. The interest in these cases is motivated by the need to understand the physics of non-stationary gasdynamic phenomena, but also by the fact that unsteady blast-wave phenomena are linked to explosion dynamics, the latter being associated with the very sudden release of chemical, nuclear, electrical or mechanical energy in a limited space. An example is seen in Figure 14 where the shock wave diffraction around a cylinder and inside a channel with a cavity, at two different time instants, are shown. In [7] we have shown that accurate prediction of the unsteady wall-pressure loads in such geometries can be obtained using high-resolution Riemann solvers on structured finite volume grids, though the structure of the various shock-wave interactions is captured better when grid-adaptation is employed [32].

Furthermore, the group has been engaged with the study of Richtmyer-Meshkov instabilities [36] and interaction of shock waves with gas inhomogeneities e.g. shock-bubble interaction [37]. In the latter case, the study is motivated by the broad area of applications in which such interactions appear as well as by the need to understand the fundamental
mechanisms associated with turbulence generation and mixing. Examples of applications include cavitation damage to human tissues during diagnostic ultrasound or lithotripsy, shock/boundary-layer interaction, supersonic and hypersonic combustion, instability of collapsing gas bubbles in liquids as well as applications in astrophysics.

Figure 14: Unsteady shock-wave diffraction around a cylinder ($M_{in} = 2.81$) [7] and inside a cavity ($M_{in} = 1.3$) [35].

Acknowledgements
Various parts of this work were supported by the following projects: (i) EPSRC/MoD (GR/L18457) in collaboration with GKN Westland Helicopters and (ii) Brite/EuRam project UNSI.

The UNSI project (Unsteady Viscous Flow in the Context of Fluid-Structure Interaction) is a collaboration between Alenia, BAe, CASA, Dasa-M (Coordinator), Dassault, DERA, DLR, FFA, IMFT, IRPHE, NUMECA, ONERA, Saab, TUB and UMIST/QMW. The project is funded by the European Union and administrated by the CEC, DG XII, Brite/EuRam, IMT (Project Ref: BRPR-CT97-0583).
References


